Digital Photogrammetric System

PHOTOMOD

Version 5.21

USER MANUAL

Project Processing

RACURS

1. Block processing **7**

2. Main windows **7**

2.1. 2D Window 7
2.1.1. Image displaying tools *9*
2.1.2. Brightness and contrast adjustment *10*
2.1.3. Saving 2D window image *10*
2.2. 3D Window 12
2.2.1. 3D window properties *14*
2.3. Navigation window 17
2.4. Layer manager 17
2.5. Status panel 20
2.6. Object list window 20
2.7. Raster map window 21

3. Stereomeasurements **23**

3.1. Stereomodes 23
3.1.1. Anaglyph glasses *23*
3.1.2. Shutter glasses *23*
3.2. Stereopair selection 24
3.3. Operating marker 25
3.3.1. Moving marker mode *25*
3.3.2. Fixed marker mode *25*
*3.3.3. **Marker = mouse mode*** *26*
3.3.4. Snap-to-ground mode *26*
3.3.5. Streamline mode *26*
3.3.6. Fixed Z mode *27*
*3.3.7. **Marker** window* *27*
3.3.8. Type of snapping *28*
3.4. Adjusting stereoimage 28
3.5. Measurements over the model 29

4. Vector objects **30**

4.1. Vector layer 31
4.2. Vector layer with classifier 31
4.2.1. Creating vector layer with classifier *31*
4.2.2. Classifier *31*
4.2.2.1. Classifier creation 34
4.2.2.2. Classifier editing 36
4.2.2.3. Import of classifier 36
4.2.2.4. Adding attributes 37
4.2.2.5. Additional attributes 40
4.2.2.6. Labels creation 41
4.2.2.7. Attaching vector objects to classifier 42
4.2.2.8. Object list 42
4.3. Vector objects creation 43
4.3.1. Vector object types *43*
4.3.2. Vector objects creation *43*
4.3.2.1. Point creation 44
4.3.2.2. Polyline creation 44
4.3.2.3. Polygon creation 45
4.3.2.4. Creating **orthogonal** vector objects 45
4.3.2.5. CAD-objects creation 45
4.3.3. Loading vector data *47*
4.3.4. Saving vector data *47*
4.3.5. Switching between stereopairs *47*
4.4. Editing of vector objects 48
4.4.1. Objects selection *48*
4.4.1.1. Selection tools 48
4.4.1.2. Selection modes 48
4.4.1.3. Selecting all objects in layer 49
4.4.1.4. Selecting objects of the same code 49
4.4.2. Point editing *49*

4.4.3. Polyline/polygon editing ... *50*
4.4.3.1. Vertex editing ... 50
4.4.3.2. Vertex adding ... 50
4.4.3.3. Polyline continuing ... 50
4.4.3.4. Converting objects to figures ... 51
4.4.3.5. Interpolating polyline ... 51
4.4.3.6. Interpolation of polyline elevations ... 51
4.4.3.7. Rounding off corners ... 52
4.4.3.8. Moving polyline ... 52
4.4.3.9. Deleting polyline ... 52
4.4.3.10. Building buffer zone ... 53
4.4.3.11. Creating profiles through vector objects ... 53
4.4.3.12. Circles around points ... 54
4.4.3.13. Project on stereomodel ... 55
4.4.3.14. Projection on TIN ... 55
4.4.3.15. Quick interpolate ... 56
4.4.3.16. Deleting pickets around line objects ... 56
4.4.3.17. Converting polygons to points ... 56
4.4.3.18. Merging point objects by attribute ... 57
4.4.3.19. Searching objects by attribute value ... 57
4.4.3.20. Collating objects ... 58
4.4.3.21. Copying to layer ... 59
4.4.3.22. Copying/pasting using clipboard ... 60
4.4.3.23. Operations with group of objects ... 60
4.4.3.24. Object type conversion ... 61
4.4.4. Curves editing ... *61*
4.4.4.1. Curves creating mode ... 62
4.4.4.2. Polylines to curves conversion ... 62
4.4.4.3. Curves to polylines conversion ... 62
4.4.4.4. Automatic smoothing adjustment ... 63
4.4.4.5. Curve segments editing ... 63
4.4.4.6. Smoothing control ... 63
4.4.5. Topology ... *64*
4.4.5.1. Segment deleting ... 64
4.4.5.2. Merging polylines ... 64
4.4.5.3. Merging polygons ... 65
4.4.5.4. Closing polyline ... 65
4.4.5.5. Unclosing polyline ... 65
4.4.5.6. Splitting polyline ... 65
4.4.5.7. Connection to point ... 65
4.4.5.8. Connection to polyline ... 65
4.4.5.9. Continuing along polyline ... 65
4.4.5.10. Closing along polyline ... 66
4.4.5.11. Topology control ... 67
4.4.6. Undoing the editing operations ... *68*

5. Creation of digital terrain model (DTM) ... **69**

5.1. Types of DEM representation ... 69

6. TIN ... **70**

6.1. Initial data for TIN building ... 71
6.2. Workflow ... 71
6.3. Preparing of base layers for TIN creation ... 71
6.3.1. Pre-regions ... *72*
6.3.2. Pickets creation ... *73*
6.3.2.1. Modes and methods for creating pickets ... 73
6.3.2.2. Grid creation ... 73
6.3.2.3. Automatic computation of points ... 76
6.3.2.3.1. Computation of points ... 76
6.3.2.3.2. Correlator presets ... 80
6.3.2.3.3. Settings of correlator preset ... 81
6.3.2.3.4. More settings of pass ... 85
6.3.2.3.5. First approximation settings ... 86
6.3.2.3.6. Computing points in distributed processing mode ... 87
6.3.2.4. Creation of pickets in pathway mode ... 88
6.3.2.5. Points editing ... 89
6.3.2.6. Filter by Z-range ... 90

6.3.2.7. Median Z filter ... 90
6.3.2.8. Filter adjacent point objects ... 91
6.3.2.9. Filter of buildings and vegetation ... 91
6.3.2.9.1. Setting up parameters of filter ... 95
6.3.2.9.2. Recommendations on using the filter ... 97
6.3.2.10. Considering operator personal difference ... 98
6.3.3. Using triangulation points ... 99
6.3.4. Opening of base layers ... 99
6.4. Building TIN ... 100
*6.4.1. **Create TIN** window ... 100*
6.4.2. Building of TIN boundaries ... 102
6.4.3. Viewing TIN ... 104
6.4.4. Saving TIN ... 105
6.4.5. Loading TIN ... 105
6.4.6. Closing TIN ... 105
6.5. TIN editing ... 105
6.5.1. TIN topology verification ... 105
6.5.2. Filtering ... 106
6.5.2.1. Simplifying ... 107
6.5.2.2. Peak filtering ... 107
6.5.2.3. Filter by Z-range ... 109
6.5.3. Rebuilding of TIN ... 109
6.6. TIN control against triangulation points ... 110
6.7. Calculating TIN area ... 111
6.8. Statistics ... 112

7. Contour lines ... 113

7.1. Building contour lines from TIN ... 113
7.2. Building contour lines from DEM ... 114
7.3. Creating smooth model contours ... 116
7.4. Saving contour lines ... 118
7.5. Loading contour lines ... 118
7.6. Operations on contour lines ... 118
7.6.1. Controlling contours intersections ... 119
7.6.2. Checking contours by points ... 119
7.6.3. Merging contours ... 120
7.6.4. Checking contours merging ... 122

8. Digital Elevation Model (DEM) ... 122

8.1. DEM building ... 122
8.1.1. Building DEM using TIN ... 122
8.1.2. Building DEM using pickets ... 124
8.1.3. Building DEM using regular pickets ... 126
8.1.4. Creating smooth model DEM ... 127
8.1.5. Creation of DTM by dense model ... 128
8.1.5.1. Building parameters of DTM by dense model ... 129
8.1.6. Batch DTM creation ... 132
8.2. DEM rebuilding by TIN ... 134
8.3. DEM recovering ... 134
8.4. DEM saving ... 135
8.5. DEM loading ... 135
8.6. DEM filtration ... 135
8.6.1. Buildings and vegetation filter ... 135
8.7. Operations on DEM ... 137
8.7.1. Merging DEMs ... 137
8.7.2. Flattening DEM fragment ... 137
8.7.3. Transposing DEM ... 138
8.7.4. Converting DEM to another coordinate system ... 138
8.7.5. Transforming DEM by a set of points ... 139
8.7.6. Filtration ... 139
8.7.6.1. Median filter ... 139
8.7.6.2. Smooth filter ... 139
8.7.7. Converting DEM into points ... 140
8.7.8. Difference DEM creating ... 140

8.7.9. Filling void cells ... *141*
8.7.10. Conversion cells to void ... *145*
8.8. DEM accuracy control ... 145
8.8.1. DEM accuracy control by TIN ... *145*
8.8.2. DEM accuracy control by vector objects ... *147*
8.8.3. DEM accuracy control by triangulation points ... *148*
8.8.4. Void cells searching ... *149*
8.8.5. DEM's comparing ... *149*
8.9. LIDAR data loading ... 149

9. Import-Export ... 149

9.1. Import vector objects ... 149
9.1.1. Coordinates conversion during import ... *150*
9.1.2. Importing attributes ... *151*
9.1.3. Import from ASCII format. ... *151*
9.1.3.1. ASCII format description ... 152
9.1.4. Import from ASCII-A format ... *152*
9.1.4.1. ASCII-A format description ... 153
9.1.5. Import from CSV format ... *154*
9.1.6. Import from DGN format ... *156*
9.1.7. Import from DXF format ... *157*
9.1.8. Import from Generate format ... *158*
9.1.9. Import from ATLAS KLT format ... *159*
9.1.10. Import from LAS format ... *159*
9.1.11. Import from LIG format ... *160*
9.1.12. Import from MIF/MID format ... *161*
9.1.13. Import from Shape format ... *163*
9.1.14. Import from VectOr format ... *164*
9.1.15. Import pickets from VectOr format ... *164*
9.2. Export ... 165
9.2.1. Coordinates conversion during export ... *165*
9.2.2. Export to ASCII format ... *166*
9.2.3. Export to ASCII-A format ... *166*
9.2.4. Export to CSV format ... *167*
9.2.5. Export to DGN format ... *168*
9.2.5.1. DGN format peculiarities ... 169
9.2.6. Export to DXF format ... *169*
9.2.7. Export to Generate format ... *171*
9.2.8. Export to ATLAS KLT format ... *172*
9.2.9. Export to LIG format ... *172*
9.2.10. Export to MIF/MID format ... *173*
9.2.11. Export to Shape format ... *175*
9.2.12. Export to VectOr format ... *175*
9.3. Import of DEM ... 176
9.3.1. Batch DEM import ... *177*
9.4. Export of DEM ... 177

10. Parameters and settings ... 178

10.1. Windows settings ... 178
10.2. Marker settings ... 180
10.3. Control settings ... 181
10.4. Settings of modules starting ... 183
10.5. Settings of block layout ... 183
10.5.1. Parameters of loading and displaying images ... *184*
10.6. Stereo settings ... 185
10.7. Correlator settings ... 186
10.8. DEM settings ... 188
10.9. TIN settings ... 189
10.10. Grid settings ... 191
10.11. Pathway mode settings ... 191
10.12. Vector objects settings ... 192
10.12.1. Settings of labels displaying ... *193*
10.12.2. Settings of elevation labels displaying ... *194*

10.12.3. Settings for display of points numbers *195*
10.12.4. Coordinates transformation *197*
10.13. Parameters of images and DEM loading 198
10.14. Orientation settings 199
10.15. Undo settings 199
10.16. Settings for resource backups 200
10.17. Auto-save settings 201
10.18. System settings 202
10.19. Mouse settings 203

11. Three-dimensional modeling – 3D-Mod 203

11.1. Main menu of 3D-Mod program 204
11.2. Texture mapping to 3D objects 209
11.2.1. Texture mapping to 3D objects *209*
11.2.2. Objects editing in 3D Mod *214*
11.2.2.1. Removing of the stereo-plotting errors of the initial 2D vector objects 214
11.2.2.2. Editing of 3D objects 217
11.3. Appendix A. Description of ASCII format 218
11.4. Appendix B. Description of ASCII-A format 218

1. Block processing

As soon as the block adjustment operation is complete (see **Block adjustment** User manual) you can start with block processing, which includes the following capabilities:

- 3D vectorization in stereo mode – (see the chapter Vector objects creation)
- Creating DTM as an irregular network of triangles **TIN** (see the chapter TIN)
- Creating DTM as a regular grid **DEM** (see the chapter DEM)
- Creating smooth contour lines with given z-interval (see the chapter Contours)
- Orthomosaicking (see the **Orthomosaic building** User Manual)

2. Main windows

There are two types of windows mainly used on block processing stage for viewing and editing DTM and other data - **2D** and **3D**

2D window is used for viewing and editing of DTM, 3D vectors and contour lines in stereo or mono modes (see the chapter 2D Window)

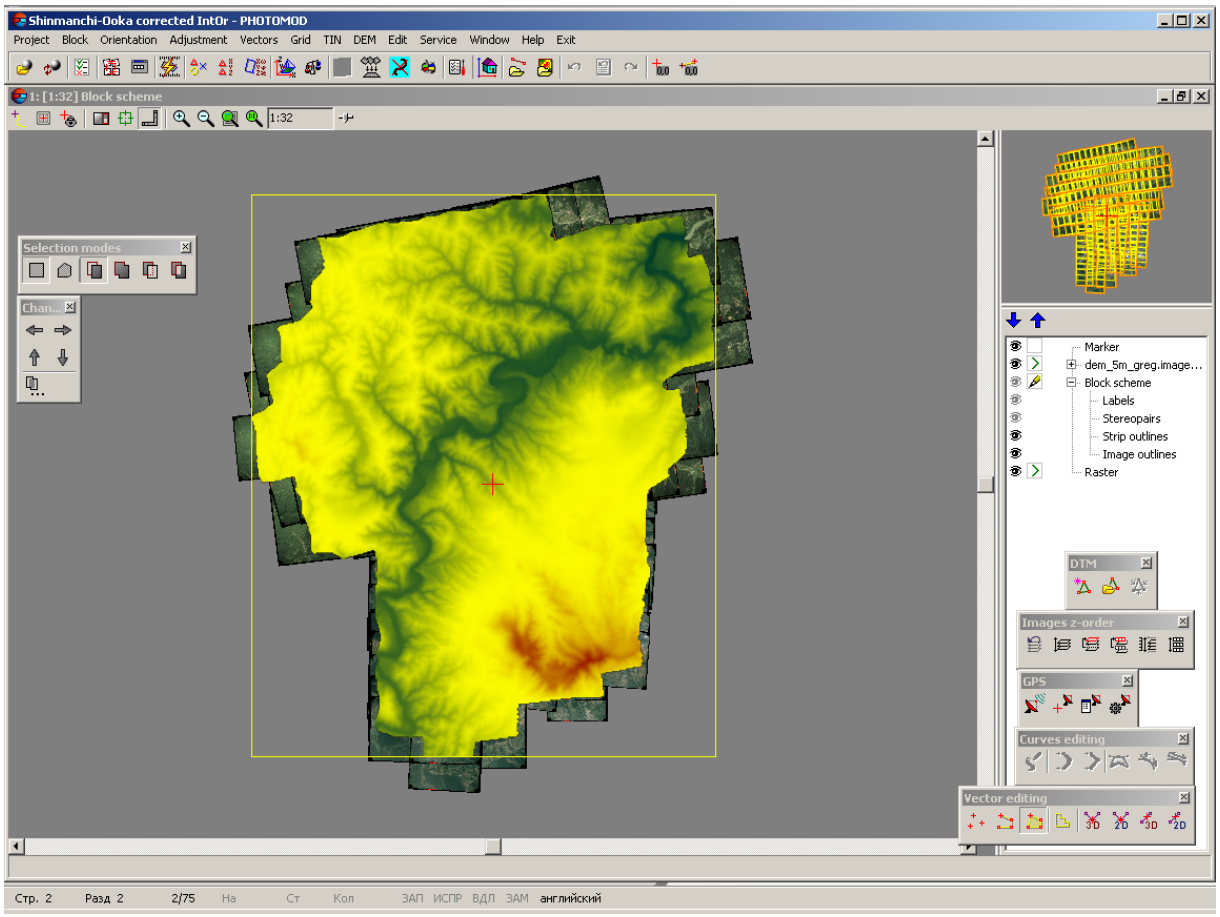
3D window is used for viewing DTM and vector objects at different angles in 3D space (see the chapter 3D Window)

According to the corresponding option on the **Windows** tab in the **Settings** window (**Service / Settings** menu command), a 2D window is opened automatically after starting **PHOTOMOD Core**.

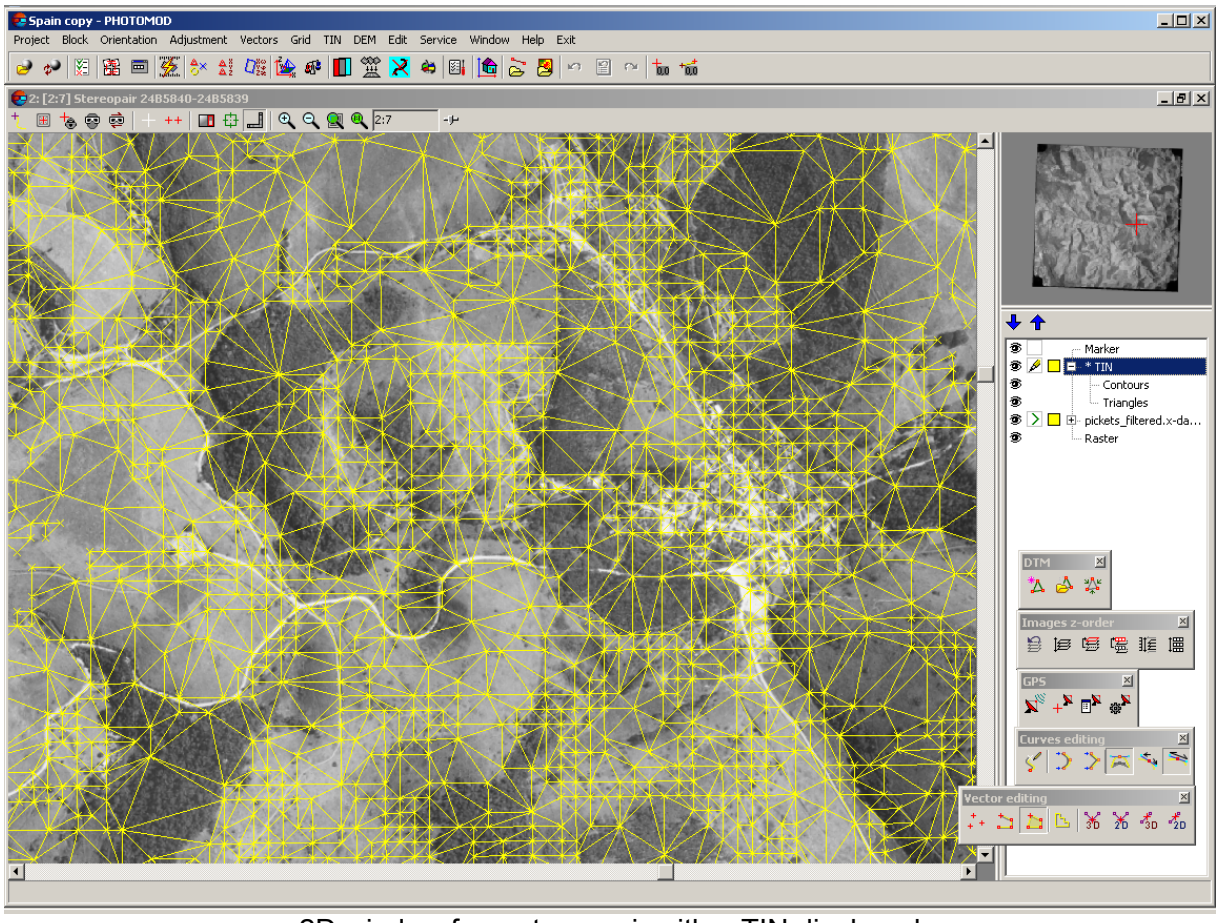
2.1. *2D Window*

2D window is the main window used for creation and editing of 3D vector objects, TIN and DEM. Once **PHOTOMOD Core** is started **2D window** displays the full scheme of the block. The window content is managed by the **Layer manager** window (see the chapter Layer manager)

There are two types of **2D windows -** a window with full block scheme and a window with a single stereopair. In order to open a new **2D window** with block scheme use menu command **Windows | New 2D window (block)**; for a new 2D windows with stereopair - **Windows | New 2D window (stereopair)** or the button on the **PHOTOMOD Core** main toolbar. Thus it is possible to work with several windows simultaneously if needed. Use command **Window | Refresh all 2D- windows** to refresh all opened 2D windows contents.



2D window with a block scheme and a block-wide DEM displayed



2D window for a stereopair with a TIN displayed

There are following icons in the 2D window tool bar:

- (**marker == mouse** / **F4** hot key) – turns on **marker == mouse mode**, in this case mouse cursor is not displayed and only the stereomarker is used for work (see the chapter marker == mouse mode).
- (**center on marker** / **F7** hot key) – scrolls image in such a way that the marker is located in the middle of screen
- (**fixed marker** / **F6** hot key) – turns on/off the mode when marker is always located in the middle of screen and while moving the mouse the image "moves" (see the chapter Fixed marker mode).
- (**fixed marker parallax mode** / **Shift+F7** hot key) – toggles fixed parallax mode. *The command is available for the selected stereopair in **2D** window.*
- (**stereo mode** / **F9** hot key) – turns on/off stereomode. You can also change the type of stereo by using (**Service | Settings | Stereo** option). Page-flipping and analog stereomodes are available. See the chapter Stereomode. Icon is available for **2D window** for selected stereopair.
- (**change phase** / **F11**) – for mono-mode loads left or right image of stereopair For steromode changes the phase - replace left and right image in the window. Icon is available for **2D window** for selected stereopair
- (**adjust depth** / **F2** hot key) – adjusts stereopicture so that the X-parallax for marker is equal to 0. See the chapter Stereoimage settings. Icon is available for **2D window** for selected stereopair.
- (**restore depth** / **F3** hot key) – cancels "adjust depth" operation (see above). See the chapter Stereoimage settings. Icon is available for **2D window** for selected stereopair.
- – shows / hides **Navigation** and **Manager** windows, located to the right from the main window (see the chapters Navigation window and Layer manager).
- – shows / hides **Navigation** window (see the chapter Navigation window)
- – (duplicated by the **Ctrl+F8**) shows / hides scrollbars

The rest of **2D window** toolbar icons are used for image displaying (zooming, panning, scrolling, etc (See the chapter Image displaying tools)

Shift+F8 hot keys open the panel used to setup image radiometric properties
See the chapter Brightness and contrast settings.

There are two common ways to select images for *opening a stereopair* in a 2D window from the Block editor or Block scheme window:

- Select two overlapping images.
- Select single image – in this case, a stereopair window is opened for the stereopair formed by the selected image and the next image in strip, if there is one – or with the previous image, if selected image is the last one in its strip.

2.1.1. Image displaying tools

Following icons of **2D window** are used to zoom in / zoom out the image:

- (* hot key) – one step zoom in;

- (/ hot key) – one step zoom out;
- (**Alt-Enter** hot key) – fit image to the window;
- (**Alt-1** hot key) – 1:1 zoom, when image cell corresponds to screen pixel.

For convenient image zooming in **2D window** use the following hot keys:

- **Alt-2** – 200% zoom
- **Alt-3** – 300% zoom
- **Alt-4** – 400% zoom

In order to change the current zoom level you can also use a slider , for selecting given zoom value. Mouse click along with **Ctrl-Alt** or **Ctrl-Alt-Shift** can be also used to zoom in and zoom out respectively.
Besides, you can zoom in by zoom box along with pressed **Ctrl-Alt** and zoom out by zoom box along with pressed **Ctrl-Alt-Shift**. For “panning” over the image move the mouse with pressed **Alt** key and left mouse button, or use the scroll bars.

Navigation window is used for fast navigating into the needed part of the image.

2.1.2. Brightness and contrast adjustment

To change image brightness, contrast and gamma use sliders , , located at the bottom of 2D window. The panel is show or hidden by **Shift - F8** hot keys.

A set of buttons located to the right from sliders is used for adjustment of image brightness, contrast and gamma for color channels (red, green, and blue). You can adjust the parameters either for each selected channel (the button corresponding to the channel color should be pushed), or for all channels at the same time (when the button is pushed).

When working in stereo mode (turned on by pushing the button on 2D window toolbar or by **F9** hotkey) these operations may be performed either for each image of stereopair (to turn on the right or the left image the button or correspondingly should by pushed), or for the whole stereopair (when the button is pushed).

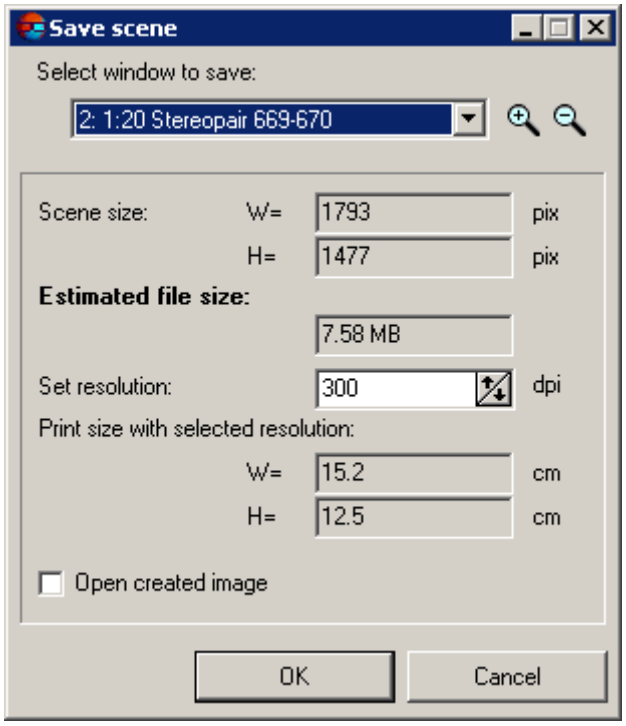
To restore default BCG settings select the command **Restore settings** in context menu of BCG panel.

These settings are kept only for current session; they are reset when either **PHOTOMOD** is restarted or the 2D window is closed.

2.1.3. Saving 2D window image

The system provides the ability to save the 2D window scenes to TIFF file format (with the pyramid). Full scene (not only the part visible inside the window) is saved, with regard to settings in the Layer manager, current zoom and images order (set with the use **Images z-order** toolbar). For the 2D window (block scheme), besides the image, the tab data file with the georeference data is also stored in the current geo coordinate system of block scheme.

Use the **Service | Save image** menu command to save the scene. The **Save scene** window is opened.



Settings for saving 2D window scene

Choose the window to save from the drop down list of all open 2D windows. and buttons are used to set needed window zoom. Set image resolution in dpi – to obtain acceptable image print size.

To open and (optionally) print the image immediately after saving, use the **Open created image** checkbox.

To save the scene, press OK button and select path and TIFF file name. At that, it saves the whole scene, not just the visible fragment in a 2D window. Note that the bigger the zoom, the larger the file and the longer it takes to save the scene.

Notes:
1. At least a small part of 2D window should be visible to save the scene.
2. In order to save stereo image, anaglyph stereo mode should be chosen (see the chapter Stereo settings). It is also possible to save either left or right frame in mono mode using the ***Change phase button of the 2D window.***

Image order in block scheme window may be adjusted with the **Images z-order** toolbar:

The toolbar contains the following tools:

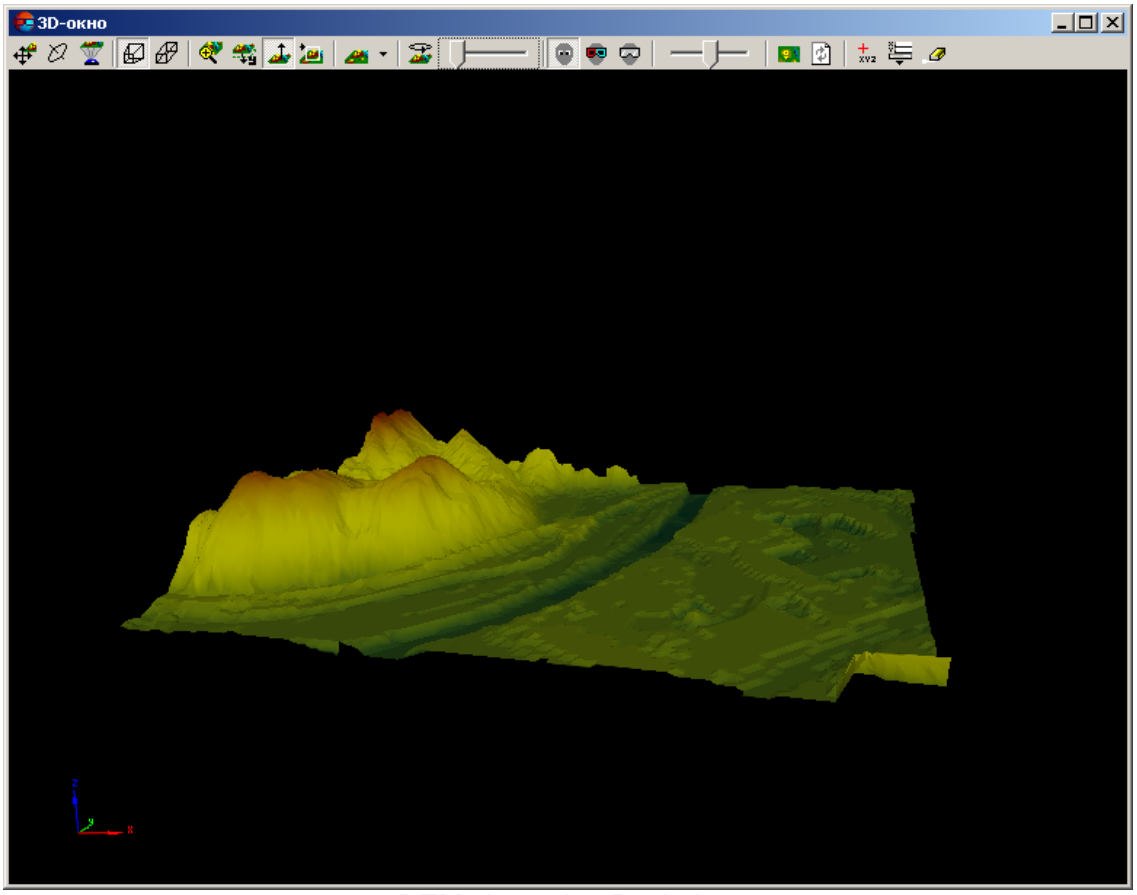
- – reset images order – set block to initial look;

- – invert images order (invert order of images in all strips, and the strips order as well;
- – bring selected images to front;
- – bring selected strips to front;
- – invert images order in selected strips;
- – invert strips order.

2.2. *3D Window*

3D window is opened by **Windows | 3D window** command and used to 3D displaying of DEM, TIN and vector objects at different angles. You can view the model in mono and anaglyph or page-flipping stereomodes. Besides moving and rotation you can change Z-scale of the model and use it for animated presentations. See also the chapters Vector objects, DEM creation.

3D window uses **OpenGL** interface for objects displaying.



DEM shown in 3D window

3D window tool bar includes following icons:

- – **Moving (Alt- left mouse button** hot keys**)**. Used to move view area by left mouse button or by arrow keys.
- – **Rotation.** Used to rotate view area by using left mouse button along with an **Auxiliary round** displayed on the screen. You can show / hide this round by pressing

on the icon by right mouse button. If the cursor is located inside the round the model is rotated around global XY axes. Otherwise - around Z axis.

- – **Camera focal length.** Used to change camera focal length.
- – **Perspective projection**. Used to view the model in perspective projection.
- – **Orthogonal projection**. Used to view the model in orthogonal projection.
- – **Scaling**. One step zoom in / zoom out (duplicated by the mouse wheel).
- – **Scene center moving**. Used to move scene center by moving the mouse along with its left button pressed..
- – **Scaling along object's Z-axis**. Used to expand / constrain model along Z-axis (duplicated by +/- hot keys respectively). The model is changed by moving the mouse along with its left button pressed.
- – **Select**. Used to select a model part by moving the mouse along with its left button pressed. Duplicated by **Ctrl** and left mouse button) The selection is used for detailed viewing
- – opens pull-down menu used to manage modes of model or its part viewing:

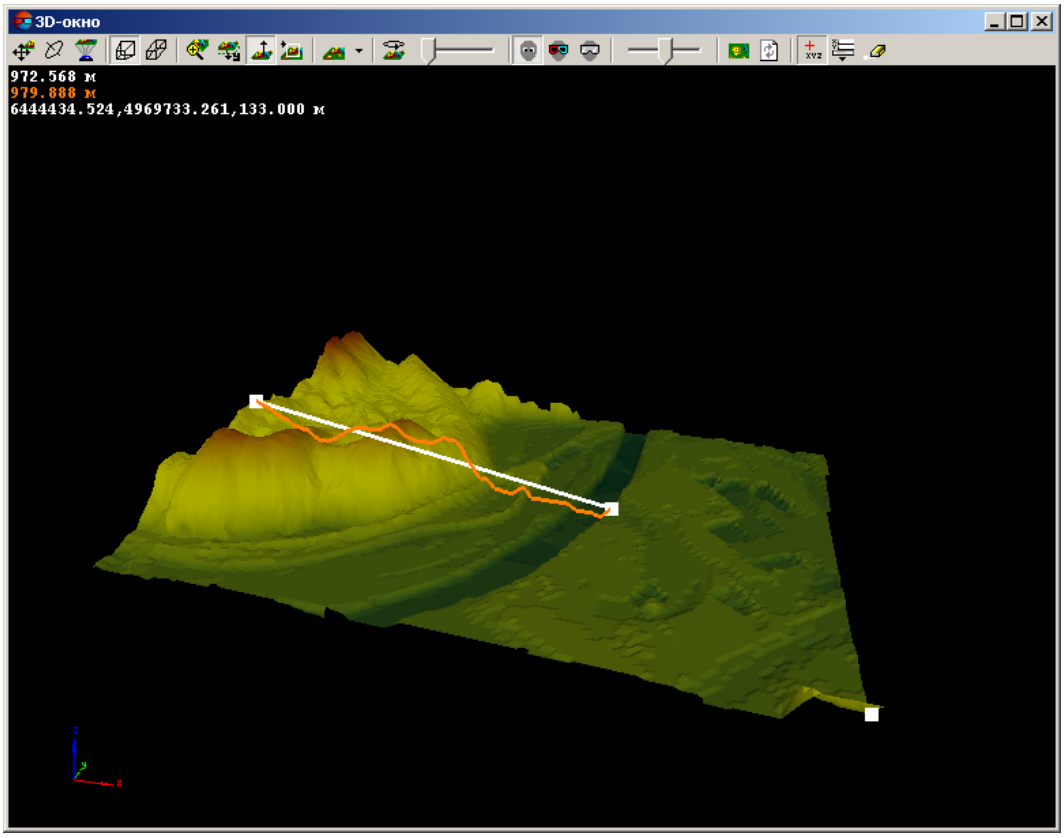
  – **Full layer** – total model displaying;
  – **Show selected** – displaying of model selected part;
  – **Points** – displaying model part as points;
  – **At least one point is selected** – displaying the selected part of model in such a way that all model elements belonging to the selection even partially will be selected (for example is selected area includes a part of vector object the total object will be selected).
  – **Fully selected** – displaying selected part of model which includes only elements fully covered by the selected area.

- – **animation** Used to view the 3D model from all sides. The slider is used to change the speed of model rotation. Right mouse button click on the icon opens pull-down menu containing the items related to rotation modes and turning on / off animation

  – **On/Off animation**;
  – **Around X-axis** – rotation around X-axis;
  – **Around Y-axis** – rotation around Y-axis;
  – **Around Z-axis** – rotation around Z-axis;
  – **Change the direction** – changing the direction of rotation;

- – **Mono**. Monomode on-off (by default).
- – **Anaglyph stereo**. Anaglyph stereomode on-off.
- – **Page-flipping stereo**. Page-flipping stereomode on-off. Slider , is used to setup the depth. Use right mouse button to change the stereo phase **(Exchange buffers for stereopair**).
- – **Default**. Used to go back to preliminary settings.
- – **Refresh**. Refreshes model state in case of changes in 2D window (automatic in perspective projection mode).
- – **Show point coordinates**. Shows coordinates of a model point picked by mouse. Besides you can measure distances in this mode. First and last point of the line are selected by double mouse click. Current cursor coordinates are shown at the upper left corner as well as the straight distance and the distance over relief. Also you

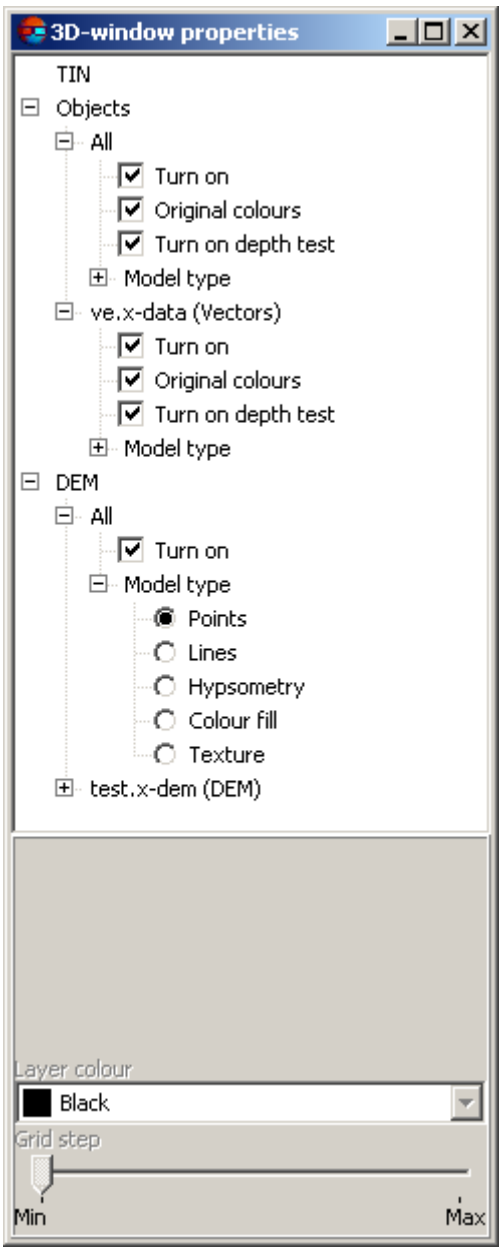
can set the first and last points coordinates by the icon. Use icon to delete the current measured line.



Measuring distances and cursor coordinates in 3D window

2.2.1. 3D window properties

3D window properties window is used to manage displaying layers. This window is opened automatically when opening 3D window.



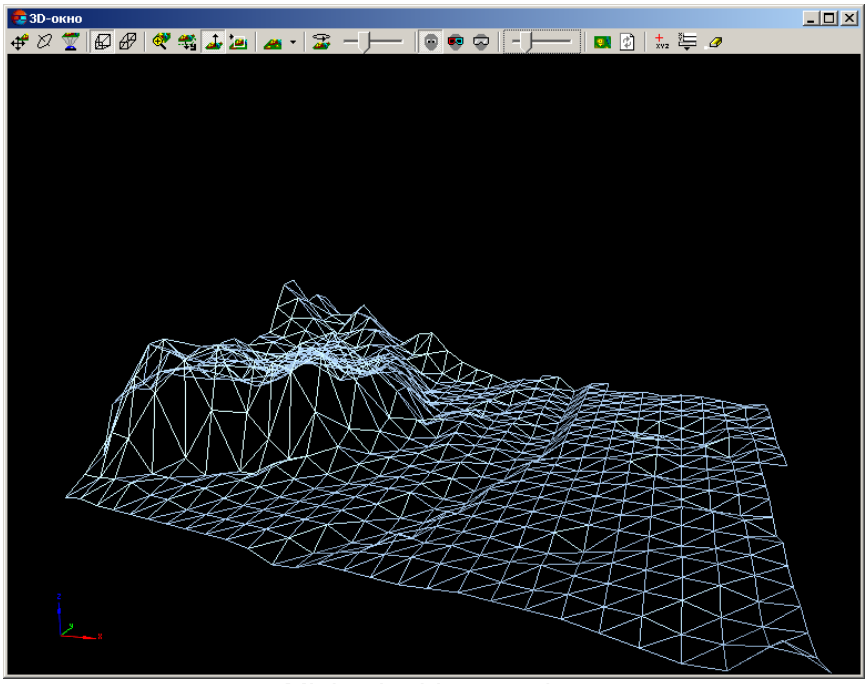
Use **On** option, to show / hide following objects and layers in 3D window, as well as for some other settings:

- **TIN** – **TIN** layer managing (see also the chapter Triangulated Irregular Network (TIN):
  All – shows / hides all TIN layers;
  Model type – TIN displaying mode:
  – **Points**;
  – **Lines -** wireframe model;
  – **Hypsometry** - hypsometrical model;
  – **Texture -** model with a rendered source raster (available when the raster is georeferenced)
  Imaging – changes layer color;
  – Shows the layer as monochrome (**Lightning**) or by colors in accordance with the Z-range (**Color fill**) When using **Lightning** mode you can change TIN color from pull-down menu **Layer color** after TIN selection;
- **Objects** – managing vector layers (See also the chapter Vector objects creation):
  All – shows/hides all vector layers and objects;

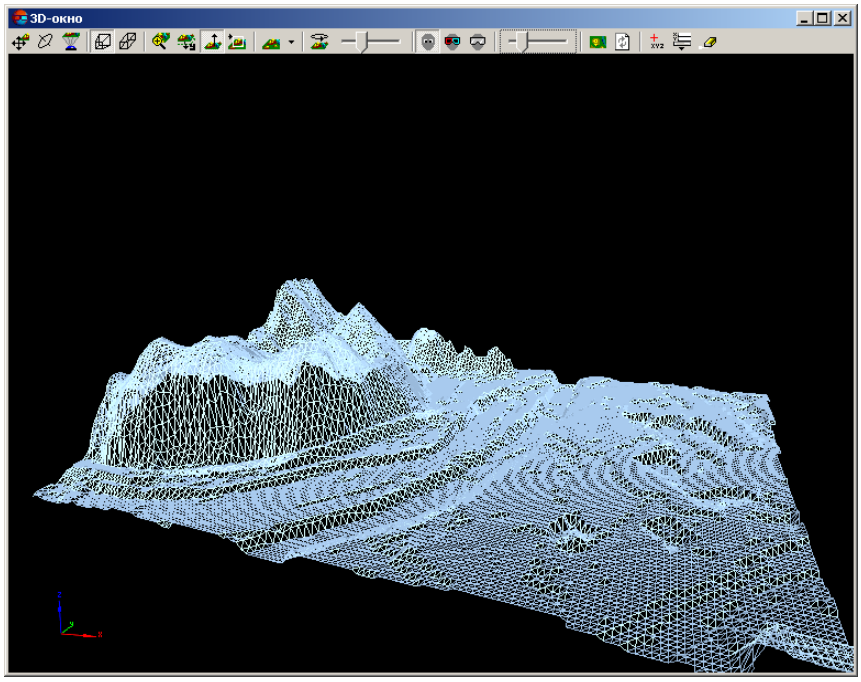
Original colors shows vector objects by colors used while the objects creation. If this option is Off, you can use colors from the **Layer color** pull-down menu located at the window bottom;
Turn on depth test shows / hides vector objects, “covered” by the model.
Model type – selects the type of vector objects displaying (**Vectors**, **Bezier**);

- **DEM –** managing DEM layer (see also the chapter DEM):
  Model type – DEM displaying mode:
  – **Points**;
  – **Lines -** wireframe model;
  – **Hypsometry** - hypsometrical model;
  – **Color fill** - the model colored in accordance with Z-coordinate;
  – **Texture -** model with a rendered source raster (available when the raster is georeferenced).

DEM and TIN color can be changed by using the color you need fron the pull-down menu (**Layer color**) in the bottom of **3D window properties** window. **Grid step** slider is used to change the level of model details. Thus use low grid step value to see the model generally and increase it to see the details of its different parts.



Minimal grid step value



Maximal grid step value

2.3. *Navigation window*

The **Navigation window** is located at the upper-right part of 2D window. It shows the full content of 2D windiow and used for fast moving over it. Click the place you need in the navigation window and 2D window will be scrolled correspondingly. Green frame in the navigation window bounds the image fragment currently displayed in 2D window

Following icons are used to manage 3D window:

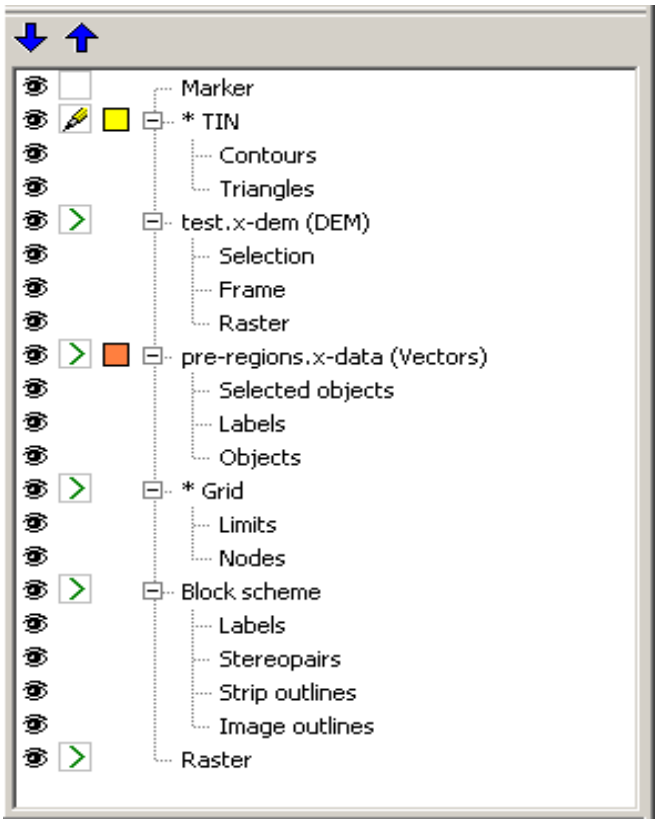
– shows / hides **Navigation window**;

– shows / hides **Navigation window** along with the **Layer manager** (see also the chapter Layer manager).

2.4. *Layer manager*

Layer manager is used to show / hide different kinds of objects and layers in **2D window** (see the chapter 2D window). **Layer manager** window is opened by icon of the main **2D window** tool bar.

Icons and are used to change the layers order in 2D window.



Manager Layer

Use icon , to show / hide following layers and objects in **2D window**:

- ***Raster*** – shows / hides block images. See the chapter **Project forming** в in **Project creation** User Manual.
- ***Block scheme*** – shows / hides the block scheme containing following objects:
  - **Names –** names of project images;
  - **Stereopairs –** stereopair outlines (red color);
  - **Strip outlines** ;
  - **Images outlines**.
- ***Triangulation points –*** shows / hides point of aerial triangulation (tie, check and ground control points). See also **Aerial triangulation** User Manual. The aerial triangulation points are displayed by using **Orientation | Show as vector objects** option, after that you can save them as an ordinary vector file and use for example for DTM creation. See also the chapter Creation of digital terrain model (DTM).
- ***Pre-regions*** – shows / hides pre-regions. Pre-regions are also the vector objects and can be edited in a standard way. See the chapter Pre-regions.
- ***Grid*** – shows / hides regular grid used for DTM creation (see the chapter Grid creation), including the following:
  - **Limits –** shows/hides grid extents (red color)
  - **Nodes –** shows/hides grid nodes (green color)
- ***TIN*** – shows / hides TIN (see the chapter TIN).
- ***DEM*** – shows / hides DEM (See the chapter DEM), including following items:
  - **Selection** – selected fragment of DEM (if any)
  - **Frame** – rectangular DEM border
  - **Raster –** DEM itself.
- ***Vector*** – shows / hides vector objects (see the chapter Vector objects creation), including following object types:
  - **Selected objects** – selected vector objects;
  - **Labels** – labels;
  - **Objects –** all vector objects belonging to the current layer.

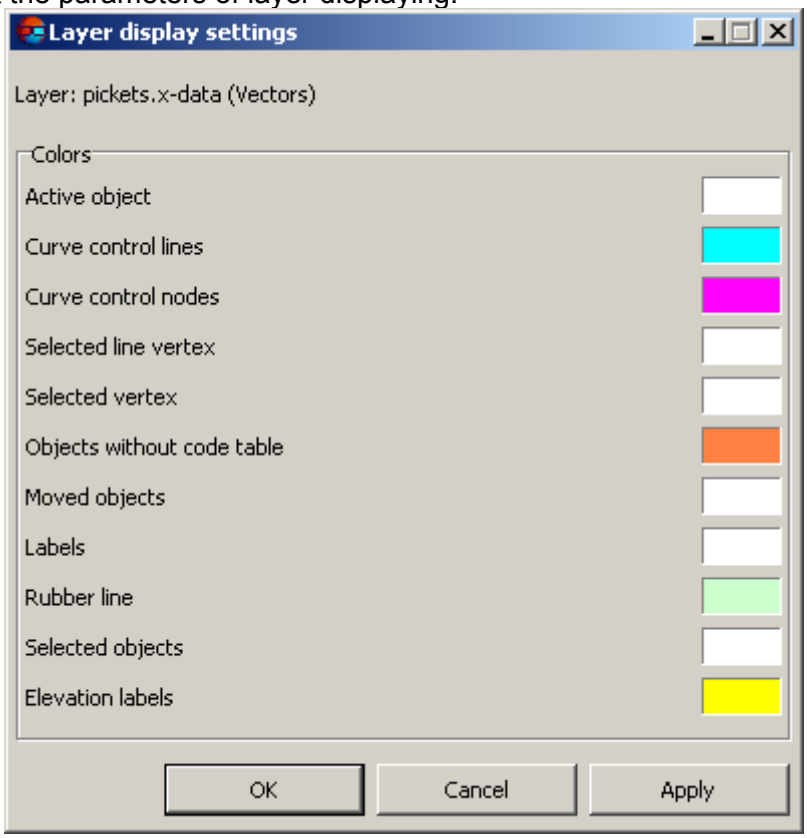
- ***Contours*** – smooth contour lines (see the chapter Contour lines). Contours layer in fact is an ordinary vector layer and consist of the same object types.
- ***Marker***

You can setup displaying parameters of different object types by using **Service / Settings**. See the chapter Parameters and settings.

Vector layers have an extension x-data, grid layers – x-grid, DEM layers – x-dem.

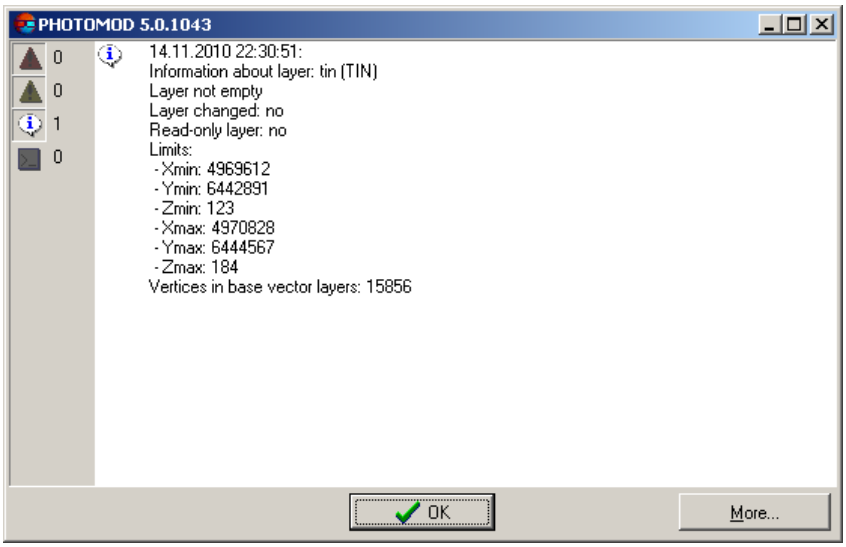
Symbol , appeared by the mouse click on the layer name makes the layer “active” and editable in **2D window**. Symbol means that layer is active but not-editable.

Symbol shows “main” color of layer object. Double mouse click on the layer name opens a window to set the parameters of layer displaying.

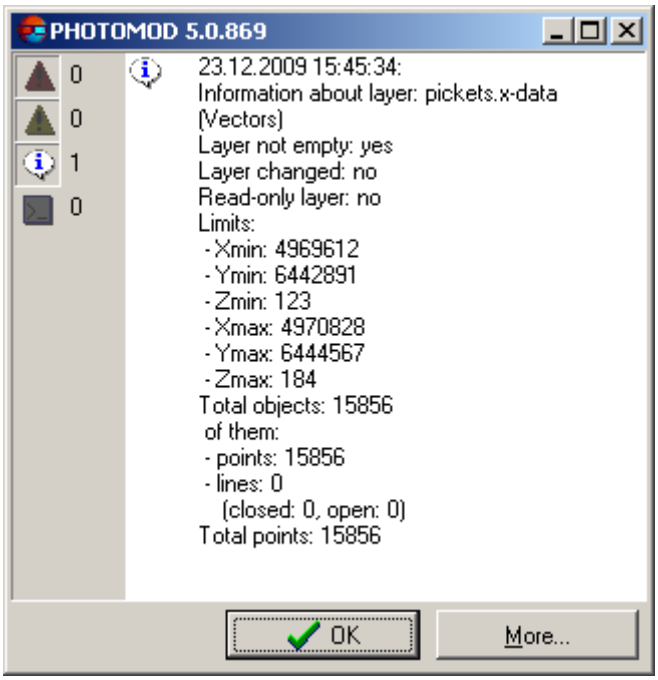


Color settings for the vector layer

Right mouse button click on the layer name opens following pull-down menu:
Information – layer related information;
Close – closing selected layer.



TIN related information



Vector pickets related information

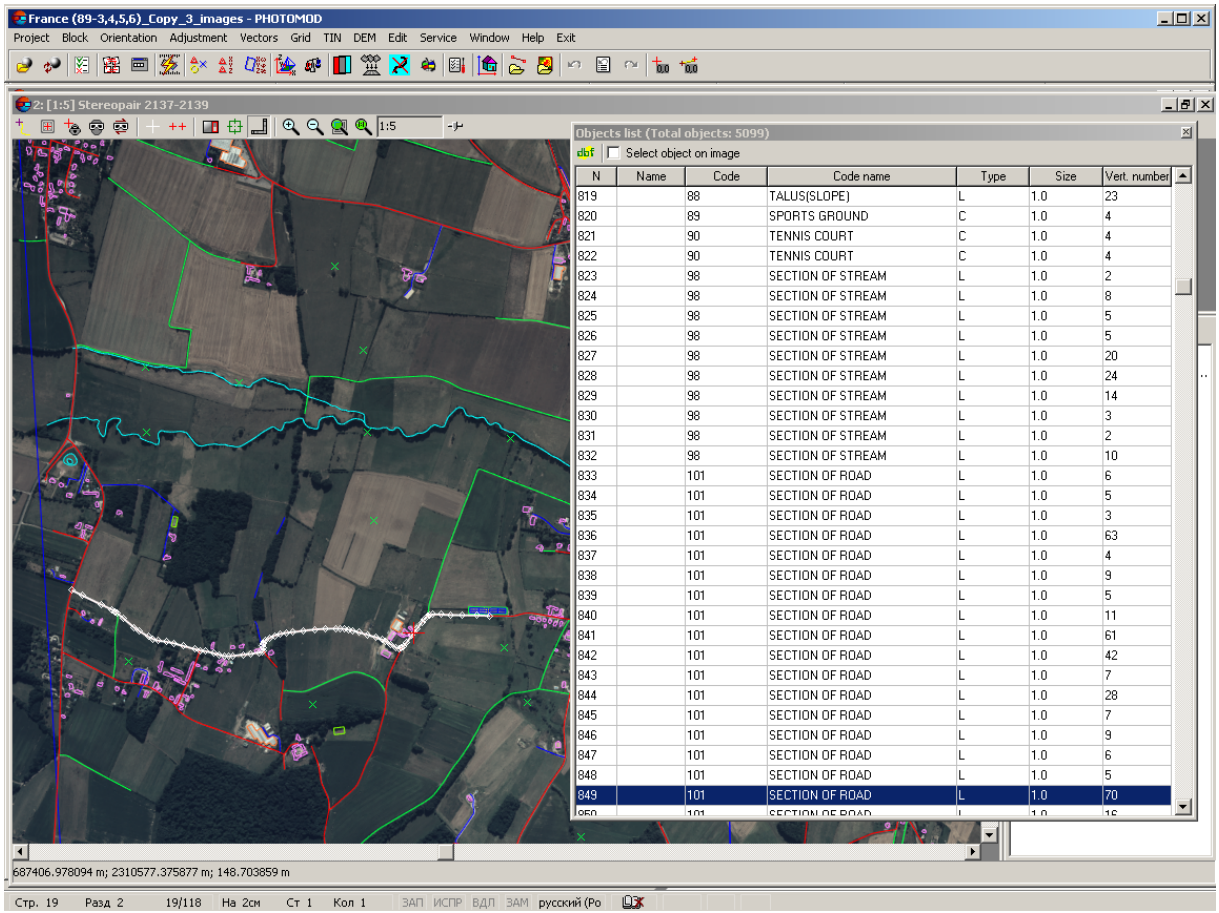
2.5. *Status panel*

Status panel is located in the bottom part of **PHOTOMOD Core** window and shows

- Current marker coordinates in the project coordinate system.
- Error messages or messages about successful operation completing. For example – «**Bad point**» or «**Ready**» when “placing” a marker on the surface by correlator (spacebar hot key).

2.6. *Object list window*

The list of vector objects loaded into a layer with classifier can be displayed in the **Object list** window (**Windows / Object list** menu).


France (89-3,4,5,6)_Copy_3_images - PHOTOMOD
Project Block Orientation Adjustment Vectors Grid TIN DEM Edit Service Window Help Exit
2: [1:5] Stereopair 2137-2139
Objects list (Total objects: 5099)
dbf Select object on image
N Name Code Code name Type Size Vert. number
819 88 TALUS(SLOPE) L 1.0 23
820 89 SPORTS GROUND C 1.0 4
821 90 TENNIS COURT C 1.0 4
822 90 TENNIS COURT C 1.0 4
823 98 SECTION OF STREAM L 1.0 2
824 98 SECTION OF STREAM L 1.0 8
825 98 SECTION OF STREAM L 1.0 5
826 98 SECTION OF STREAM L 1.0 5
827 98 SECTION OF STREAM L 1.0 20
828 98 SECTION OF STREAM L 1.0 24
829 98 SECTION OF STREAM L 1.0 14
830 98 SECTION OF STREAM L 1.0 3
831 98 SECTION OF STREAM L 1.0 2
832 98 SECTION OF STREAM L 1.0 10
833 101 SECTION OF ROAD L 1.0 6
834 101 SECTION OF ROAD L 1.0 5
835 101 SECTION OF ROAD L 1.0 3
836 101 SECTION OF ROAD L 1.0 63
837 101 SECTION OF ROAD L 1.0 4
838 101 SECTION OF ROAD L 1.0 9
839 101 SECTION OF ROAD L 1.0 5
840 101 SECTION OF ROAD L 1.0 11
841 101 SECTION OF ROAD L 1.0 61
842 101 SECTION OF ROAD L 1.0 42
843 101 SECTION OF ROAD L 1.0 7
844 101 SECTION OF ROAD L 1.0 28
845 101 SECTION OF ROAD L 1.0 7
846 101 SECTION OF ROAD L 1.0 9
847 101 SECTION OF ROAD L 1.0 6
848 101 SECTION OF ROAD L 1.0 5
849 101 SECTION OF ROAD L 1.0 70
687406.978094 m; 2310577.375877 m; 148.703859 m
Стр. 19 Разд 2 19/118 На 2см Ст 1 Кол 1 ЗАП ИСПР ВДЛ ЗАМ русский (Ро

Selecting object in 2D window and in the list

Following objects are displayed in the **Object list** window: object numbers, codes, types, attributes. If **Select object on image** option is On the mouse click on the record in the object list selects the corresponding object on the image.

Object list can be saved to .dbf file by using icon in the upper part of the window

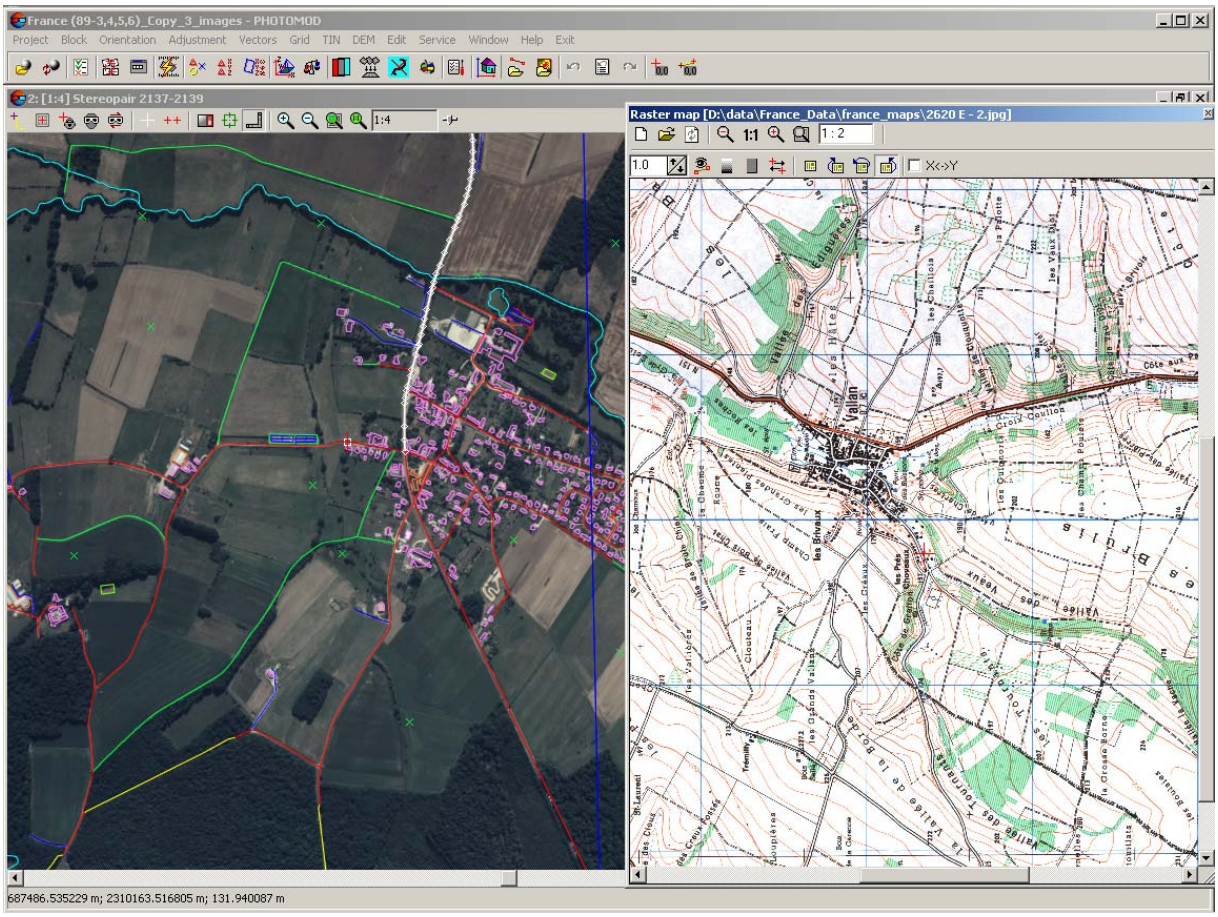
2.7. *Raster map window*

While the stereovectorization process you can load a map in a raster format which can help for objects type identification.

Raster map window can be shown / hide by using **Windows | Raster map** option.

Any georeferenced raster image could be opened in the window in the following formats: GeoTIFF, ERDAS Imagine, PCIDSK, and also TIFF, BMP, JPEG, NITF, GIF, JPEG2000, accompanying by georeferencing text files PHOTOMOD GEO, ArcWorld (TFW, BPW, JPW,…) or MapInfo TAB.

The initial raster image is georeferenced using known ground control points in **Georeference** window of **PHOTOMOD Montage Desktop** module, see an appropriate User Manual.


France (89-3,4,5,6)_Copy_3_images - PHOTOMOD
Project Block Orientation Adjustment Vectors Grid TIN DEM Edit Service Window Help Exit
2: [1:4] Stereopair 2137-2139
Raster map [D:\data\France_Data\france_maps\2620 E - 2.jpg]
687486.535229 m; 2310163.516805 m; 131.940087 m

Raster map window

In the upper part of the window there is button bar with the following buttons:

- close map

(duplicated by **Ctrl-O** hot keys) - open raster map – opens file selecting dialogue. If the map to be opened is not located within the stereopair you will get a warning

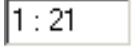
(duplicated by **F5** hot key) - refresh vectors – synchronize vectors on the raster map and stereopair

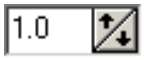
(duplicated by **/** hot key) - zoom out

(duplicated by **Alt-1** hot keys) - 1:1 zoom, when image cell corresponds to screen pixel

(duplicated by * hot key) - zoom in

(duplicated by **Alt-Enter** hot keys) - fit to page

 - zoom scale

 - relative thickness of the vector objects – the field is used for vector lines thickness changing. The value in this field is a coefficient by which absolute object thickness taken from Classifier is multiplied. Color and thickness of vectors shown on the raster map are taken from Classifier

- show vectors – to view vector objects (opened in 2D window) with raster map in background

- grayscale – transfer color raster image into grayscale

- no raster – closes raster image but preserves vector objects if any

- synchronize windows – synchronizes marker movement in 2D window and raster image window. At that when marker is moving in one of these windows it is moving in another too.

In most cases stereopair orientation is not the same as coordinate system orientation (on raster map). That is why for more convenient work the map could be rotated in the following ways: - without rotation, - turn at 90 degrees, - turn at 180 degrees and - turn at 270 degrees. At that raster file is not changed and the rotation is executed “on the fly”.

All view settings, path to loaded map, and also **Raster map** window visibility, size and location are saved automatically and restore at next **PHOTOMOD** session. At that map file name is associated with stereopair name and thus different stereopairs will be opened with appropriate raster maps.

3. Stereomeasurements

3.1. *Stereomodes*

To switch between stereomodes (**Anaglyph** or **Page-flipping**), described below use **Service | Settings | Stereo** menu. Open 2D window for selected stereo pair using the button of the main **PHOTOMOD Core** tool bar or use the **Window | New 2D-window (stereopair)**. The icon (**Open new 2D-window for selected stereopair**) of the main PHOTOMOD tool bar is used to open a stereopair for stereoprocessing.

To turn ON/OFF stereomode in 2D window click the icon in upper menu of 2D window or **F9** hot key. See the details on hardware settings of your PC for convenient working in stereomode in **PHOTOMOD Overview** User Manual.

3.1.1. Anaglyph glasses

Anaglyph stereoimage is formed by visualization of the left and right images of the stereopair “beyond” red and blue filters. To view such a picture you should use special anaglyph spectacles with red and blue glasses. Anaglyph stereomode requires no special equipment but it is not completely good for working with color images. Another disadvantage is that the picture gets a bit darker when viewing through the filters.

Note. Anaglyph stereo is available only for HighColor or TrueColor display mode of your monitor

3.1.2. Shutter glasses

Shutter glasses are liquid crystal glasses synchronized with the vertical refresh rate of the monitor. **PHOTOMOD** system supports page-flipping stereomode when working with shutter glasses. Refer to **PHOTOMOD Overview** User Manual for the details about using stereo glasses and other special equipment for stereoprocessing.

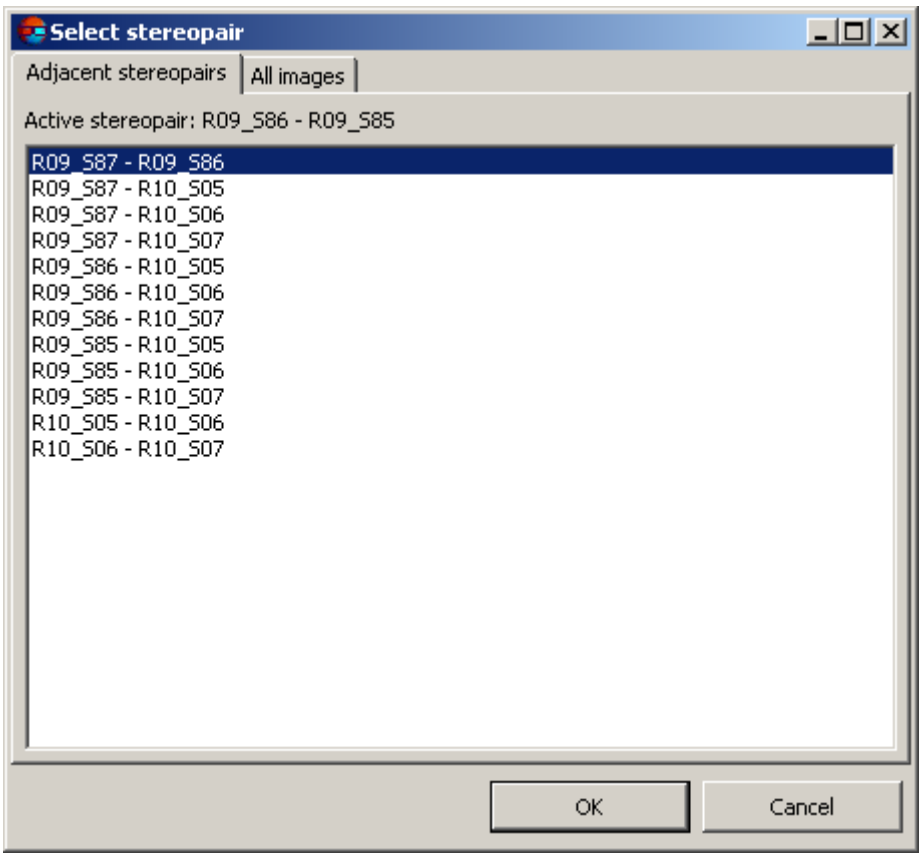
Page flipping (“frame by frame”) display mode provides the most high quality stereo picture because it uses full frames instead of semi-frames. Left and right images of the stereopair are displayed one by one synchronously with the frames switching. The shutter glasses are

synchronized with the monitor vertical refresh rate and allow you to see them "simultaneously" and make stereo measurements. For working in page-flipping mode you should use a monitor with a good enough vertical refresh rate (at least 120 Hz) and an appropriate video card.

3.2. *Stereopair selection*

To open new stereopair **2D-window** select images which make a stereopair using block scheme or **Block editor** window. If only one image is selected the stereopair composed of the selected image and the left adjacent one in the strip (if the last image of the strip is selected, right adjacent image will be opened) will be opened in the stereopair **2D-window**. Execute the menu command **Windows | New 2D window (stereopair)** or press the button on the main toolbox **PHOTOMOD Core**. Stereopair 2D window will open.
The following ways to change the stereopair are provided in the stereopair 2D-window:
– toolbox **Change stereopair** (menu **Windows | Toolbars | Change stereopair**), which contains following tools:

- – open stereopair one image backward in the strip (duplicated by the menu command **Windows | Previous stereopair**).
- – open stereopair one image forward in the strip (duplicated by the menu command **Windows | Next stereopair**).
- – open stereopair one strip upward (duplicated by the menu command **Windows | Stereopair up**).
- – open stereopair one strip downward (duplicated by the menu command **Windows | Stereopair down**).
- – select an arbitrary stereopair (duplicated by the menu command **Windows | Select stereopair**). It opens arbitrary stereopair selection window, where on the Tab **Adjacent Stereopairs** the list of all possible adjacent stereopairs is displayed (including stereopairs composed of non-adjacent images or images from different strips provided that they are overlapping), the tab **All images** displays the list of all images in the project. The images of the opened stereopair are checked. Select the stereopair using one of the tabs and press OK.



Arbitrary stereopair selection

The menu command **Windows | Open inverted stereopair** allows swapping opened stereopair images (upper down, lower up).

3.3. *Operating marker*

3.3.1. Moving marker mode

Moving and fixed marker modes are turned on/off by the icon (**fixed marker**) or **F6** hot key.
Use **mouse** or **keys with arrows** to move the marker in XY plane and **PgDn**, **PgUp** keys or the mouse wheel to move it along Z axis.
Use **spacebar** key to place the marker on the model surface automatically by the correlator. If the correlator fails a message **Bad point** appears in the Status bar accompanied with a sound (see the chapter Correlator parameters).

Note that the step of marker moving along Z axis is discrete and defined directly by the current zoom level. For fast marker moving along Z axis use mouse wheel along with pressed **Alt** key.

Use **Service | Settings | Marker (stereopair)** window to setup marker shape, size and color, see the chapter Marker settings.

3.3.2. Fixed marker mode

Moving and fixed marker modes are turned on/off by the icon (**fixed marker**) or **F6** hot key. In case of fixed marker the marker is always located in the center of screen and its X parallax is equal to 0.

During vectorization you move the images of stereopairs by the **mouse** or **arrow keys** (←, ↑, →, ↓) in plane and by **PgDn**, **PgUp** keys or the **mouse wheel** by Z. Use **spacebar** key to place the marker on the model surface automatically by the correlator. If the correlator fails a message **Bad point** appears in the Status bar accompanied with a sound (see the chapter Correlator parameters).

This mode is familiar for operators experienced in working with analytical stereo devices. Another advantage is a smooth roam vectorization with a constant image auto scrolling.

Use **Service | Settings | Marker (stereopair)** window to setup marker shape, size and color, see the chapter Marker settings.

3.3.3. Marker = mouse mode

This mode (turned on by clicking the icon **marker == mouse** or **F4** hot key) "removes" mouse cursor from the screen. In this case any mouse movement causes the corresponding stereomarker moving without mouse click. The mode is useful for digitizing polylines and polygons.

Use **Service | Settings | Marker (stereopair)** window to setup marker shape, size and color, see the chapter Marker settings.

3.3.4. Snap-to-ground mode

Marker may automatically follow the relief elevation in **Snap to ground** mode during vectorization. The snap-to-ground mode is toggled with toolbar button of the **Vectors** toolbar, **T** hotkey or **Edit | Snap to ground** menu command.

In this mode, the operator moves the marker in XY-plane, and marker height is adjusted automatically by the correlator (see the chapter Correlator settings). If correlation in the given point fails, the marker may be positioned manually with mouse wheel or **PgUp**, **PgDn** keys.

3.3.5. Streamline mode

Objects in 2D window may be created by freehand drawing with left mouse button pressed, adding points automatically when distance from the previous point exceeds the given threshold (see the chapter Settings of vector objects displaying). The mode is toggled by the button on **Vectors** toolbar, **Y** hotkey or **Edit | Streamline mode** menu command. The first point of the line is created with Insert key, further points are added automatically when either the marker is moved with left mouse button pressed, or each time the mouse button is clicked, given the above-mentioned threshold condition.
In order for this mode to work, the line or polygon drawing mode must be chosen (line (L) or polygon (C) code selected in the classifier in case it is used).

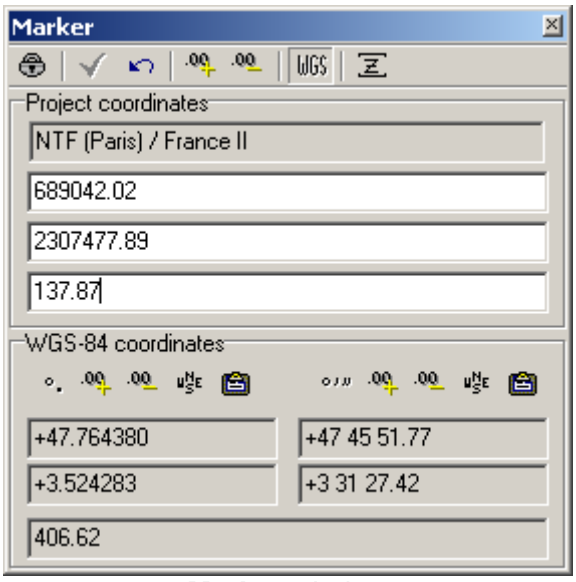
3.3.6. Fixed Z mode

If you need to draw a vector line at a constant Z level use **Fixed Z** mode. To set a Z value, place the marker to a correct position and select menu command **Edit | Fix marker by Z** or press **Alt-Z** shortcut or click the icon in the **Marker** window opened by the icon of the main panel or by the command **Windows | Marker window**. You can also enter the Z value in **Z** field of **Marker** window in order to move marker appropriately.

This mode is useful for example for digitizing the contour lines

3.3.7. **Marker** window

Marker window (opened by the icon **Marker window** of the main panel or by main menu command **Windows | Marker window**) shows marker coordinate in the project coordinate system and in WGS 84 latitude-longitude. Besides viewing the values you can enter them from the keyboard and the marker will be moved accordingly after pushing the **Apply** button.



Marker window

There are the following icons at **Marker** window button bar:

- - **apply immediately** – change marker position right after entering coordinate values without pressing **Enter** key
- - **apply** – moving marker in accordance with entered coordinate values
- - **canceling** coordinate values input
- - **more decimal places** – increasing number of decimal places in coordinate values at 1
- - **less decimal places** – decreasing number of decimal places in coordinate values at 1
- - **fix marker by Z** – Duplicated by **Alt-Z** shortcut. In this case Z-marker coordinate can not be changed. See the chapter Fixed Z mode.

Marker latitude / longitude on the WGS 84 ellipsoid are displaying at the bottom part of the window. There are flowing icons in there:

- – **coordinate display format** - changing the format of latitude-longitude values
- – **increase display precision** – more decimal places in coordinate values
- – **decrease display precision** – less decimal places in coordinate values
- – **format for units and hemisphere display**
- – **copy coordinates to clipboard**

3.3.8. Type of snapping

When working in the **snap** mode the marker is moving only “along” the existing vector objects (points, vertices or segments). It is useful when you need to create an object that spatially coincides with some existing objects. For example when you vectorize electric power line connecting existing piers (point objects). Snapping can be applied to the vector objects belonging to all layers displaying on the screen (not only to the active one). Use options of **Edit / Snapping** menu to switch between following snapping modes:

- **3D snapping to vertex** (the

icon or **V** hot key). In this mode the marker “jumps” from one vertex to another. When you click somewhere on the image, the marker moves to the nearest vertex or point.
- **2D snapping to vertex** (the

icon or **B** hot key). In this mode the marker “jumps” from one vertex to another and XY marker coordinates coincide with XY coordinates of the vertex. Marker Z coordinate at that will be preserved.
- **3D snapping to line** (the

icon or **N** hot key). In this mode the marker moves along existing vector objects lines (segments) keeping all XYZ coordinates. When you click somewhere on the image, the marker moves to the nearest vector object.
- **2D snapping to line** (the

icon or **M** hot key). In this mode the marker moves along existing vector objects lines (segments) just in XY plane. Marker Z coordinate at that will be preserved.

2D snapping is used when the creating object must spatially coincide with an existing one only in XY plane. For example you want to “draw” an extension to some building at the other Z level.
Hot keys listed above are used for quick setting snapping **on** (the key is pressed down) and **off** (the key is released).

Note: It is possible to draw part of the object (for example, part of the polyline) on the existing object if the snapping mode is turned on during the object creation. As a result, two (or more) of the object will have the geometrically coincided parts of objects.

3.4. *Adjusting stereoimage*

No matter which kind of stereomode you use, you need to adjust the stereo image to get the best stereo effect in some particular image region. This is especially important when you work with “deep” images, i.e. those having big variations in X parallax. To adjust the stereo image, do the following:

- place the stereo marker on some point of the region of interest manually or by snap to ground operation (see the chapter Operating marker);

- press **F2** key or click the icon of **2D window**.

The program will rebuild the stereo image to set X parallax in the point of stereo marker to zero value. So the best stereo picture is in the area around the marker.

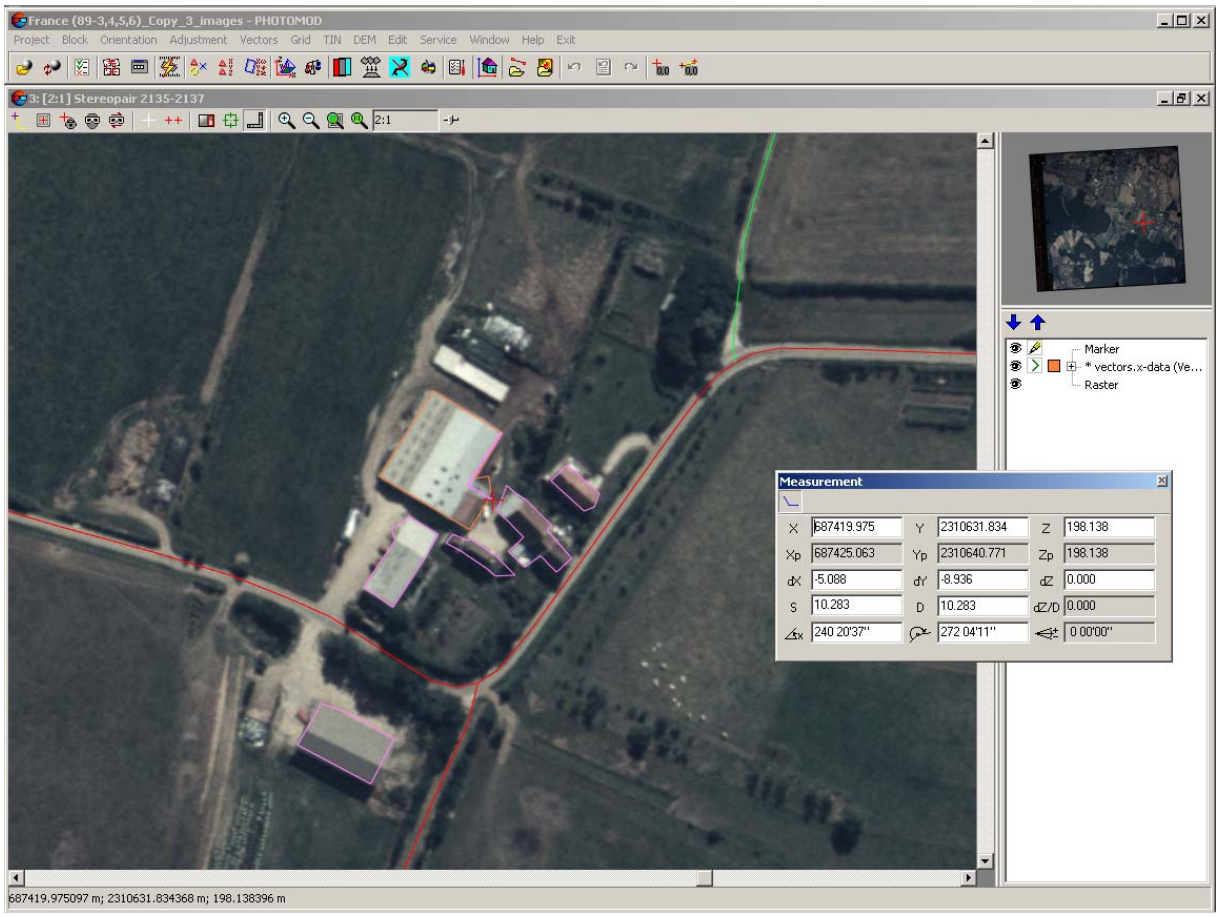
Press **F3** key or click the icon of the main panel to restore the stereo depth.

You can change stereomode phase (i.e. swap over left and right images) using **F11** key or the button in 2D window.

Besides, you can use **Shift-PgUp/PgDn** or **Shift-mouse wheel** shortcuts to change stereo image depth.

3.5. *Measurements over the model*

The mode of measurements (turned on by pushing the icon of the main panel or by selecting menu item **Windows | Measure mode**) is useful for some temporary stereo-measurements – when for example you need to know the height of a building or a tree. There are exactly the same tools as for the 3D line creation – **Insert** key to add a vertex, **PgUp**, **PgDn** keys or mouse **wheel** to move marker by Z and the mouse or **arrow** keys to move marker in plane. The line appeared is a temporal one: it disappears after you quit measurements mode (by releasing the same icon or turning off the menu item).
See also the chapter Vector objects creation.



Measurements window

Measurements window displays following values:

- **X** – marker X geodetic coordinate
- **Y** – marker Y geodetic coordinate
- **Z** – marker Z geodetic coordinate
- **Xp** – marker geodetic X-coordinate in previous point
- **Yp** – marker geodetic Y-coordinate in previous point
- **Zp** – marker geodetic Z-coordinate in previous point
- **dX** – marker geodetic delta X when drawing a “rubber line”
- **dY** – marker geodetic delta Y when drawing a “rubber line”
- **dZ** – marker geodetic delta Z when drawing a “rubber line”
- **S** – length of the current segment of the “rubber line”
- **D** – length of the current segment projection on XY plane
- **dZ/D** – the value of the current segment slope
- – direction of the current segment relatively to X axis
- – direction of the current segment relatively to the previous one
- – vertical angle of the current segment

If necessary you can input geodetic coordinates X, Y or Z into appropriate fields of **Measurement** window and marker will move into the point with specified coordinates after pressing **Enter**.

4. Vector objects

Vector objects (points, lines or polygons) are contained in a vector layer of two possible types: vector layer and vector layer with classifier.

4.1. *Vector layer*

A vector layer containing vector objects that have no attributes attached. In order to create vector layer use command **Vectors | Create layer**. As a result a new layer called **Vectors** appears in the **Layer manager** panel. Vector layer has a number and an extension “x-data” See also the chapter Layer manager.

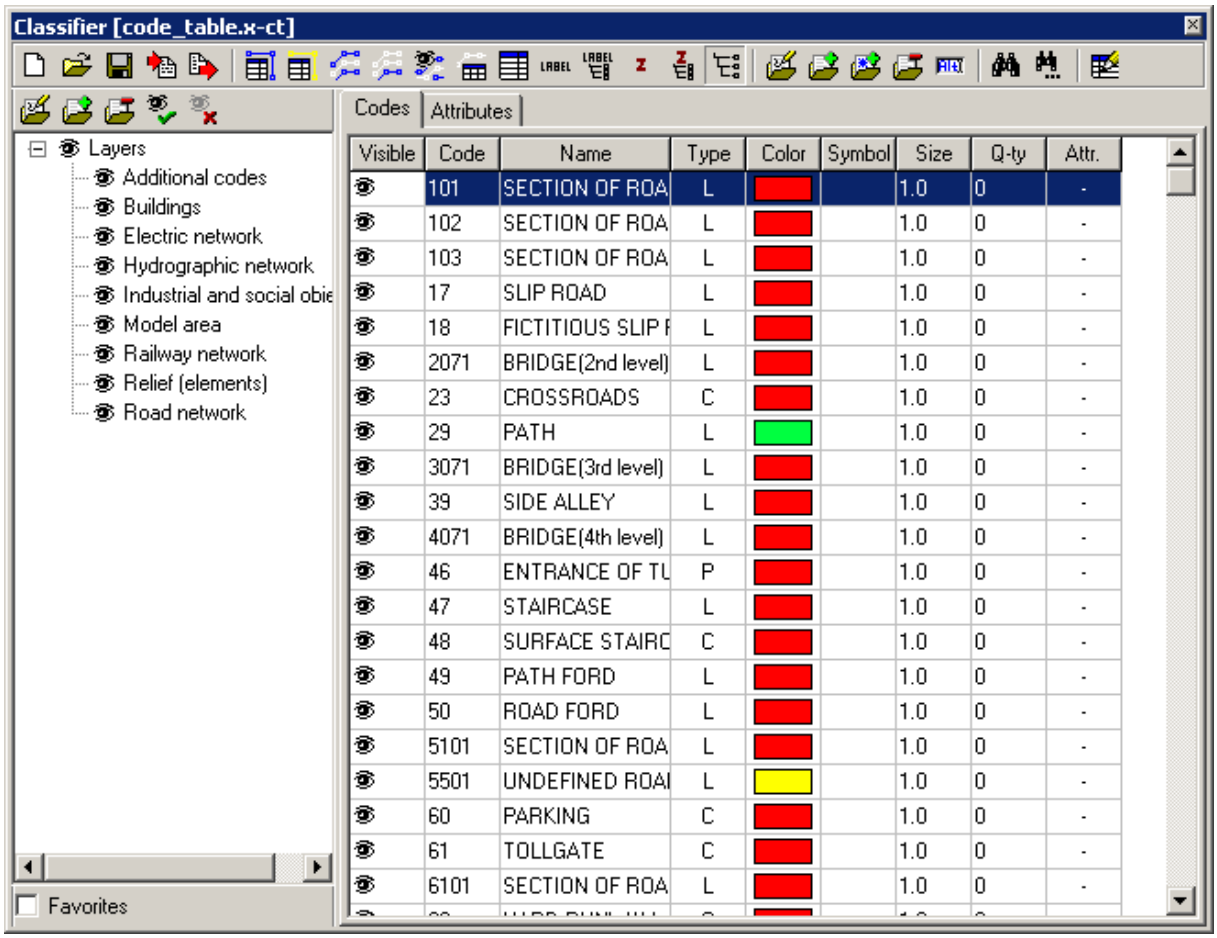
4.2. *Vector layer with classifier*

4.2.1. Creating vector layer with classifier

Vector layer with classifier contains vector objects attached to PHOTOMOD hierarchical classifier (in fact it is kind of database table). In order to create vector layer with a classifier you should use **Vectors | Create layer with classifier** command. As a result As a result a new layer called **Vectors** appears in the **Layer manager** panel. Vector layer has a number and an extension “x-data”. The classifier window will be opened automatically. See also the chapters Layer manager and Classifier

4.2.2. Classifier

All vector objects created in the vector layer with a classifier are attached to the classifier records. The classifier is a database table, containing codes and attributes used to thematic classification of the vector objects. Use **Windows | Classifier** option to open **Classifier** window.


Classifier [code_table.x-ct]
Layers
Additional codes
Buildings
Electric network
Hydrographic network
Industrial and social obie
Model area
Railway network
Relief (elements)
Road network
Favorites
Codes
Attributes
Visible
Code
Name
Type
Color
Symbol
Size
Q-ty
Attr.
101 SECTION OF ROA L 1.0 0 -
102 SECTION OF ROA L 1.0 0 -
103 SECTION OF ROA L 1.0 0 -
17 SLIP ROAD L 1.0 0 -
18 FICTITIOUS SLIP F L 1.0 0 -
2071 BRIDGE(2nd level) L 1.0 0 -
23 CROSSROADS C 1.0 0 -
29 PATH L 1.0 0 -
3071 BRIDGE(3rd level) L 1.0 0 -
39 SIDE ALLEY L 1.0 0 -
4071 BRIDGE(4th level) L 1.0 0 -
46 ENTRANCE OF TU P 1.0 0 -
47 STAIRCASE L 1.0 0 -
48 SURFACE STAIRC C 1.0 0 -
49 PATH FORD L 1.0 0 -
50 ROAD FORD L 1.0 0 -
5101 SECTION OF ROA L 1.0 0 -
5501 UNDEFINED ROAI L 1.0 0 -
60 PARKING C 1.0 0 -
61 TOLLGATE C 1.0 0 -
6101 SECTION OF ROA L 1.0 0 -

Classifier window

There is a list of thematic layers of the current project in the left part of the window. The right window part contains codes list of vector objects (**Codes** tab) and objects attributes (**Attributes** tab).
Codes tab contains the table with the following columns:

- **Visible** – visibility of objects with given code in 2D window
- **Code** – object code
- **Name** – the unique text string up to 64-character length created to refer to a vector object or group of vector objects. For example “Unpaved road”
- **Type** – one of the following types of an object: **P** – point, **L** – polyline, **C** – polygon
- **Color** – the color used to show the object on the raster image
- **Size** – the floating point number that determines the size of a vector object in ground units. For points this feature controls the size of the symbol on a raster image;
- **Symbol** – the ASCII symbol, corresponding to the object code (for point objects only), it could be selected from the standard symbol library
- **Q-ty** – quantity of objects of given code

Attr. – option of availability and number of object attributes (see the chapter Adding attributes

Upper icons of the **Classifier** window are intended for the following operations:

- - **new** – creates new Classifier after closing the loaded one (see the chapter Classifier creation)
- - **open** – opens existing classifier
- - **save as** – saves changes

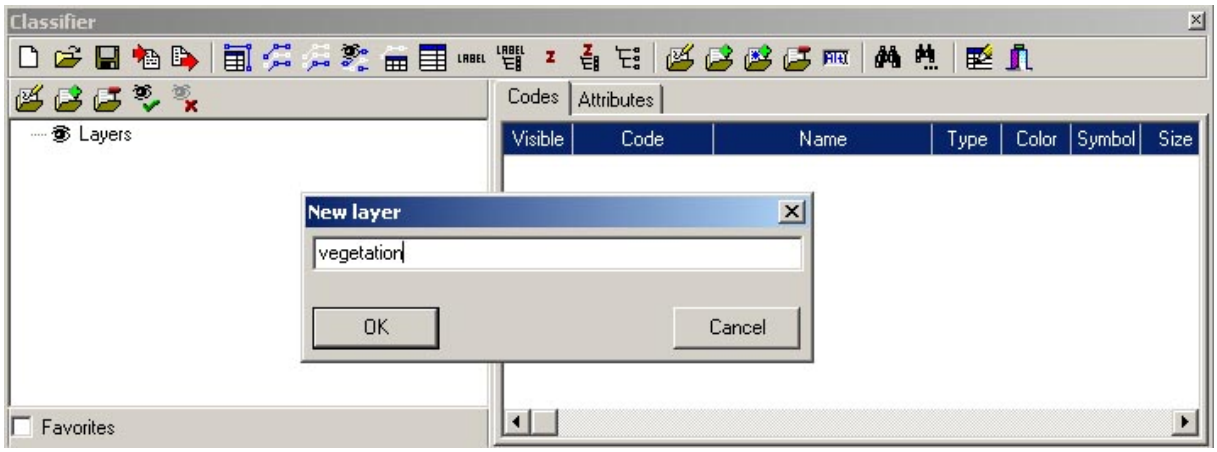
- - **classifer import** – opens standard dialogue for import codes to the current **Classifier** (see the chapter Classifer import)
- - **classifier export** – opens standard dialogue to input file name (with *.rsc extension) for export classifier to VectOr format
- - **assign code to selected objects** – allows to change the code of the objects selected in 2D window (see the chapters Attaching vector objects to classifier records и Changing object types)
- - **select all objects with current code** (see the chapter Selecting objects with current code)
- - **select all group objects** (see the chapter Selection of all layer objects)
- - **select all visible objects** – if visibility of some objects is OFF (by mouse click on the "eye" icon rightward from code or layer name) you can select the rest visible objects using this button. Use such selection to export a group of all selected objects using menu command **Objects | Import/Export | Format | Export selected objects**
- - **show selected object code** – allows to show object code in Classifier when an appropriate vector object is selected in 2D window
- - **show selected code** – used to scroll down the list of codes in the Classifier in such a way that the code of the object selected in 2D window is shown in the Classifier. This operation is useful when selected code is at the bottom of the long Classifier list and is invisible without scrolling down the list
- - **set label for current code** (see the chapter Labels creation)
- - **set label for all group codes** (see the chapter Labels creation)
- - **assign elevation from attribute to selected code** – if after vector objects import their heights are located in Classifier attributes, you can assign these heights to object vertices, by pushing this button. The operation is used during contours import from MIF/MID and SIT (Map 2005, VectOr) formats
- - **assign elevation from attribute to selected layer** – if after vector objects import their heights are located in Classifier attributes, you can assign these heights to vertices of all objects in selected layer, by pushing this button. The operation is used during contours import from MIF/MID and SIT (Map 2005, VectOr) formats
- - **show codes from sublayers** – if pushed and some layer in Layers list is selected then all vector objects codes available on this layer is shown in codes list
- - **edit code…** – opens a window for editing selected code or attribute (see the chapter Classifier editing)
- - **add code…** – opens a window for new code or attribute creation (see the chapter Classifier editing)
- - **add code to favorites** – adds selected code to another codes list, which simplifies searching for frequently used codes. Favorites list is visualized when the option **Favorites** in lower left corner of the Codes table is ON
- - **delete** – allows to delete selected code or attribute after some warning

- - **select hot key for current code** – opens an additional window which allows to associate chosen hot key (**Shift-1**,**2**,**3**,**4**,**5**) to the current vector object code, that makes easy quick code call in the Classifier
- - **find text** – opens a window of searching record in selected mode: **By code**, **By code name**. Type the value you need in the selected text field and push **OK** to start the search
- - **search again** – further code search in selected mode
- - **default classifier** – sets the currently loaded classifier as the default classifier, loaded automatically when a new layer with classifier is created.

The Classifier is a useful tool for vector objects systematization. For example you can select objects with the same code and delete them or show them with some particular color. Vector objects codes are using also for vectors export to exchange formats see the chapter Export of vector objects

4.2.2.1. *Classifier creation*

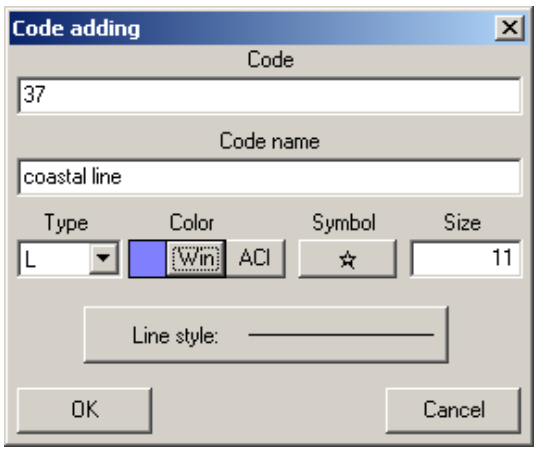
To create a classifier select menu command **Windows | Classifier**, then click the icon (**New**) in the **Classifier** window. You should start with layers creation (left part of the window)



Creating layers in the classifier

After creating the list of layers of the objects, create the list of vector objects included to each layer along with their codes and attributes.

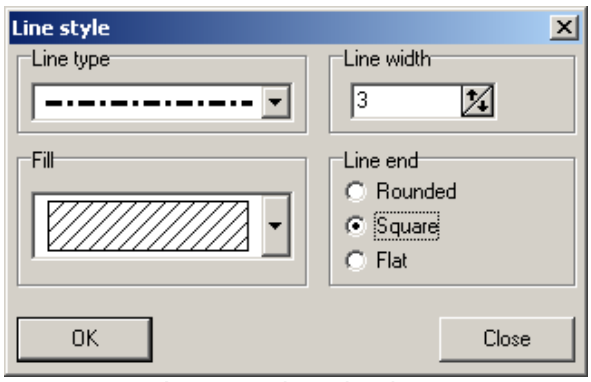
Pushing the icon (**Add code…**) for adding records to the classifier (in the right part of the window) opens the following window:



New code creation

Use **Code** field to input the unique object code, then **Code name** field for object name and also assign to the object its type and following features to display it in 2D window:

- For point object (P type) – symbol (from popped up standard symbol library) symbol color (from popping up standard MS Windows palette (**Win** button) or from AutoCAD palette (**ACI** button)) and size (in point units in the appropriate field). Please note that symbol size could be changed only for True Type fonts.
- For polyline (L type) and polygon (C type) – line color (from popped up standard MS Windows palette (**Win** button) or from AutoCAD palette (**ACI** button)) and its style in the window appearing by pushing the appropriate button:



Layer style selection

Here select line type, style of its filling, its width and shape of its end using appropriate options. Selected line style is preserved during line objects export into DXF and DGN formats, see the chapter Export of vector objects.

When all necessary options are set, push **OK** and object code will be added to the layer which name is selected in the **Layers** list or to the root directory if the objects are not divided into layers.
So you can create codes for different objects which supposed to be extracted and add them to the Classifier with possible classification into thematic layers.

Once the **Classifier** is created save it using the icon (**Save as**) to the resource with specified name. The classifer will be opened automatically for all **PHOTOMOD** further sessions. If you need to open another classifier click the icon (**Open**) to get a list of available resources with classifier. Also you can create new classifier using an imported one (see the chapter Classifier import).

Besides, while importing vector objects to PHOTOMOD from exchange formats their codes could be also imported into a new classifier or added to the existing one depending on the appropriate options, see the chapter Import of vector objects.

4.2.2.2. *Classifier editing*

In open **Classifier**, you can edit **Layers list** in the left panel and **Codes** and **attributes list** in the right one. **Layers list** editing is described in the chapter Adding attributes.

Use the tool bar above the **Layers list** for the following operations:

- (duplicated by **F2** key) **edit layer name…** – allows to change selected layer (folder) name. Press **Enter** to save changed name or **Esc** to cancel editing
- **add layer…** – opens a window for new layer (folder) name input
- **delete layer** – deletes selected layer (folder) from Layers list

Use the same buttons of upper button bar for Classifier editing in the right part of window:

- **show layer with sublayers** – shows objects in layer including those in sub-layers
- **hide layer with sublayers** – hides objects in layer including those in sub-layers
- **edit code…** – (or double click on the code record selected) opens a dialogue described in the chapter Classifier creation
- **add code…** – opens a dialogue used to create a new record in classifier
- **add code to favorites** – adds selected code to another codes list that simplifies searching for frequently used codes. Favorites list is visualized when the option **Favorites** in lower left corner of the Codes table is ON
- **delete code** – deletes selected classifer record

Popup menu opened by right mouse button click on **Layer list** allows to perform the following operations with the list:

- **Copy layer** – copies the layer selected to the clipboard
- **Paste layer** – pastes the layer copied from clipboard into Layers list
- **Move layer** – moves the layer taken from clipboard into Layers list or inside selected layer (as a sublayer) and removes it from the previous location
- **Show layer with sublayers** – opens in 2D window objects with codes, which belong to the active layer (the icon leftwards from layer name becomes active at that)
- **Hide layer with sublayers** – hides in 2D window objects with codes, which belong to the active layer (the icon leftwards from layer name becomes inactive at that).

4.2.2.3. *Import of classifier*

In order to import the classifier use the icon (**Classifier import**) which opens the dialogue for file selection. There are several possibilities of vector objects codes import when creating the classifier or adding codes to the current one:

- import of **PHOTOMOD VectOr** classifier (.rsc files). **PHOTOMOD VectOr** uses a library of map symbols (classifier) to display and print out digital map objects. The set

of standard classifiers (.rsc files) related to different map scales is included to **PHOTOMOD** distribution file set and stored in \VectOr\RSC directory by default. Use the **PHOTOMOD** classifier, corresponding to the VectOr selected classifier to associate the objects created in **PHOTOMOD** with the appropriate symbols in **PHOTOMOD VectOr**. In that case the table code will be equivalent to the classifier code for the exact object type. If you want to automatically create a PHOTOMOD classifier based on the **VectOr** classifier you should select type of file as **RSC** and select source classifier in the **Import** window.

- import of **PHOTOMOD VectOr** classifier, associated with existing map (.sit and .map files) to **PHOTOMOD** classifier (see also the chapter Import from VectOr).

After selecting the file to be imported define the name and path for the output resource with classifier in **PHOTOMOD** format.

Besides, while importing vector objects to PHOTOMOD from exchange formats their codes could be also imported into a new classifier or added to the existing one depending on the appropriate options, see the chapter Import of vector objects.

4.2.2.4. *Adding attributes*

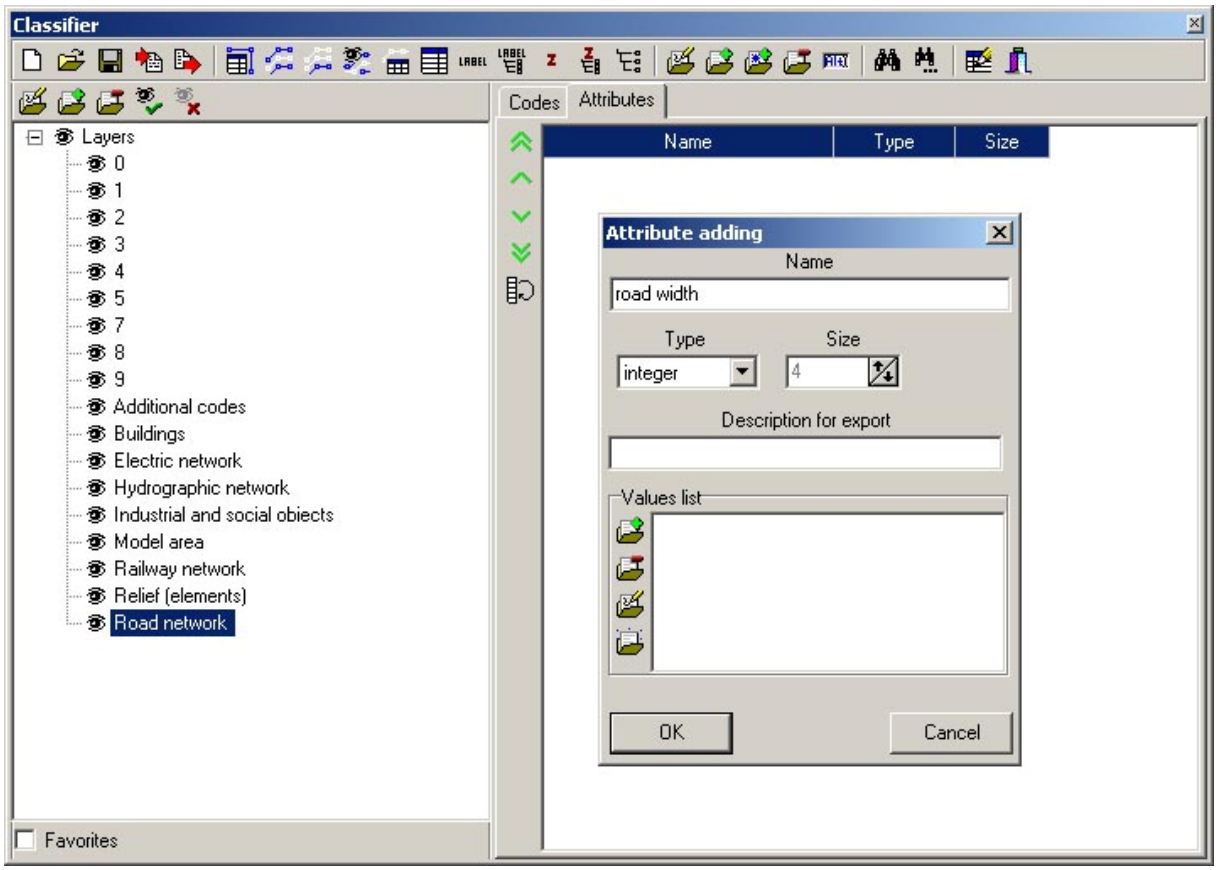
For each vector object code you can add some auxiliary attributes using the **Attributes** tab in the **Classifier** window with the following parameters:

- **Name** – field name
- **Type** – field type (see below)

Text	- text attribute either without parameters (in this case the length of the attribute is the length of its value, but not greater than 32767) or with one parameter, specifying the fixed length of the text;
Integer	- 4 byte integer (without parameters);
Smallint	- 2 byte integer (without parameters);
Decimal	- fixed decimal point attribute with two parameters. The first parameter is the total attribute length in bytes, the second - the number of digits after the decimal point;
Float	8 bytes float attribute (without parameters);
Data	unformatted attribute with one parameter specifying the attribute length in bytes;
Logical	logical attribute with possible values 0 or 1 (without parameters).

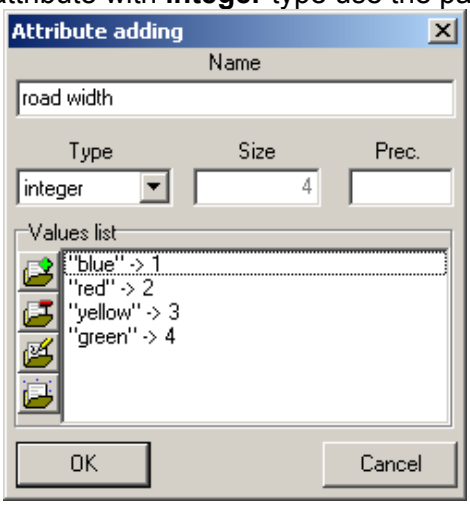
- **Size** (bytes) – for numeric fields is defined automatically, otherwise should be defined by user (for text fields for example)
- **Precision** – number of decimal places for “float” attributes

To edit or create attribute table click the icon (**Edit attribute**) of the upper button bar in **Classifier** window, when **Attributes** tab is selected. After that you will get the window for editing the parameters of attributes of record selected:



Adding the attribute

To assign attribute value to attribute with **integer** type use the panel **Values list**:



Following icons are used to:

- **add** – opens a window for input of attribute name and value

- **delete** – deletes attribute value from the list

- **edit** – opens a window for editing of attribute name and value

- **clear** – deletes all attributes values from the list.

Besides, on **Attributes** tab you can use the following operations:

- - **add attribute** – opens a window for creating a new attribute record

- - **delete attribute** – deletes current attribute field

Use the buttons on the left side of **Attributes** tab to perform the following operations with object's attributes:

- **move up** – moves the record to the topmost position

- **upper** – moves the record on one position up

- **lower** – moves the record on one position down

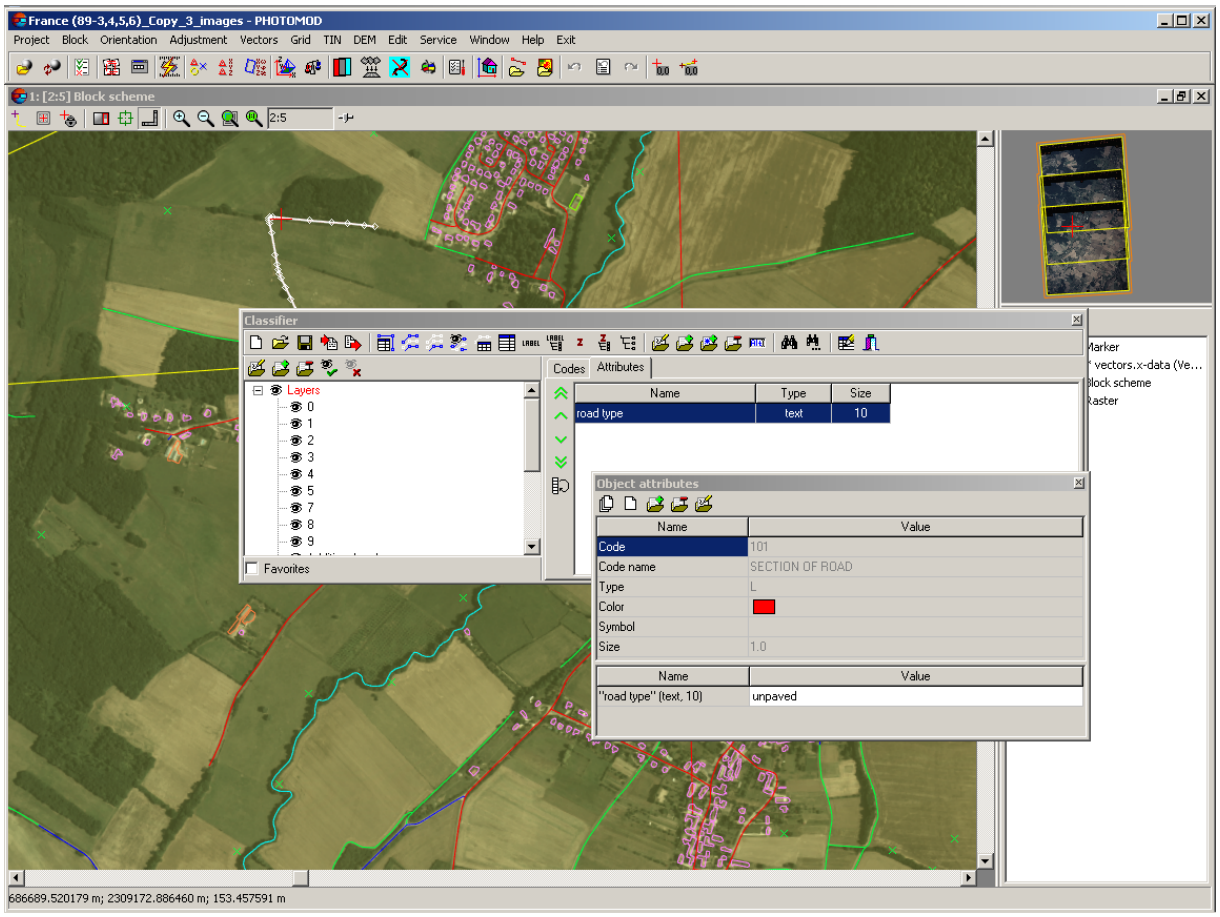
- **move down** – moves the record to the bottommost position

- **invert attributes order** – rearranges the list in the reverse order.

On the **Attributes** tab of classifier use popup menu opened by right mouse button click, which contains the following commands:

- **Copy attribute** – copies selected attribute to the clipboard
- **Copy all code attributes** – copies all attributes of the current code to the clipboard
- **Paste attribute** – pastes attribute(s) from the clipboard to the attributes list of the current code. It is used to remove unnecessary attributes from other codes of the Classifier. If you need to move attributes from one Classifier to another, paste the attribute from clipboard to text file. Then open another Classifier, select the code and paste the attribute into code selected.
- **Get Z from attribute** – assigns Z coordinate to the objects of selected code. The coordinate is either taken from imported file or input by the user.

To enter the attribute values use the command **Windows | Objects attributes**. Select **Attributes** tab in the **Object attributes** window appeared and enter attribute values in the **Values** column after clicking the grey field.



Vector object attributes

To start editing the attribute value in table cell, select it by mouse. To complete editing and save the input value, press **Enter** or just click the next cell with attribute value. Press **Esc** to cancel editing and exit.

The following operations are available for attribute's values:

- **Copy to clipboard** – allows to copy attribute's values of the object to the clipboard using menu command **Edit | Objects | Copy attributes values** if the object is selected in the current layer
- **Paste from clipboard** – object's attribute value, which was copied to the clipboard by previous operation, could be assigned to the one or several selected objects of active vector layer. Use menu command **Edit | Objects | Paste attributes values** to add attribute's values to the attributes with the same names.

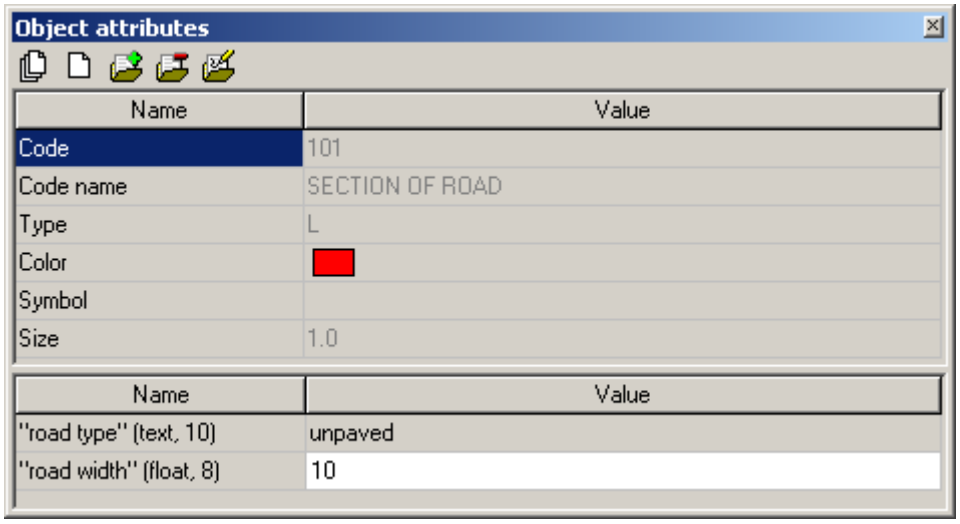
In **Object attributes** window you can create an additional attributes for the objects selected, using **Additional attributes** tab (see the chapter Adding additional attributes).

4.2.2.5. *Additional attributes*

Besides the attributes associated with a code in the classifier (see the chapter Adding attributes) the unique attributes for the selected object could be created. At that all objects associated with some code in the classifier will have common attributes defined in the classifier, but each of them may have some additional unique attributes. To create a table of additional attributes you should do the following:

- select vector object or group of objects (see the chapter Objects selection)
- select menu command **Windows | Object attributes**. First two tabs of **Object attributes** window show a record in the classifier and object attributes

- select the **Additional attributes** tab and create a table of additional attributes in exactly the same way as for classifier attributes (see the chapter Adding attributes).
  Following icons are used to create additional attributes table:
  - - **delete all additional attributes for selected objects** – deletes all additional attributes for selected objects
  - - **delete same additional attributes for selected objects** – deletes additional attributes which are common for selected objects
  - - **add additional attribute** – opens a window for creating a new additional attribute
  - - **delete additional attribute** – deletes current additional attribute
  - - **edit additional attribute** – opens a window for editing current additional attribute
- select the fields by the mouse and enter the attribute values to associate the additional attributes with the selected vector object.



Additional attributes

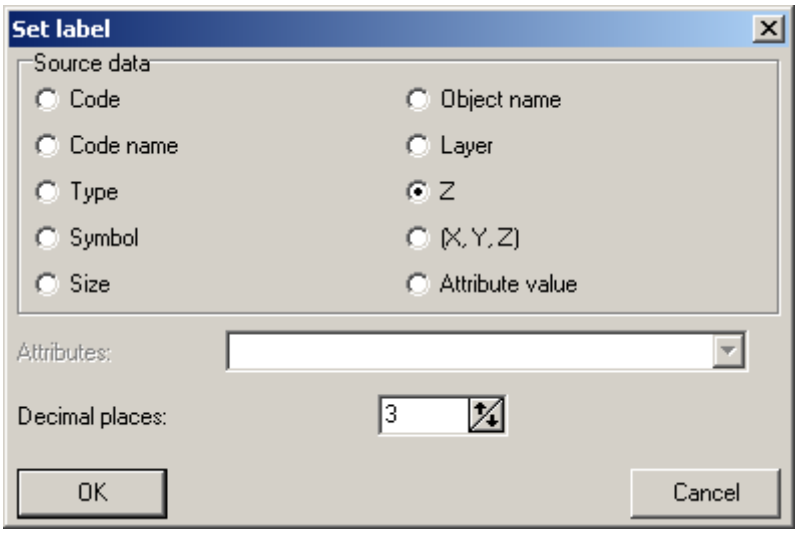
To start editing the attribute value in table cell, select it by mouse. To complete editing and save the input value, press **Enter** or just click the next cell with attribute value. Press **Esc** to cancel editing and exit.

Note. Additional attributes could not be exported in PHOTOMOD VectOr module unlike usual object attributes

4.2.2.6. *Labels creation*

Each vector object could be supplied with special sign (label) displayed in 2D window as a label to the point object or to the linear object vertex (if the appropriate option in module settings is ON, see the chapter Labels displaying settings). This label could be also exported to DXF file of AutoCAD system. To assign the label, select object code in **Classifeir** and push the icon

(**set label for current code**) in the upper button bar.



Label creation

In the list appeared choose the label source (taken from the values of columns of the layer selected in codes list). If **Attribute value** is selected, **Attributes** pop down list is available to select the label content, which will be visualized in 2D window or AutoCAD program after export of the objects of selected code to DXF file.

If you need to assign a label to all objects of selected layer push the button

(**set label for all group codes**). At that the label is assigned to all objects in selected layer, that is why you should assign it only after creation of all layer objects just prior their export to DXF.

After vector objects export to DXF format (menu command **Vectors | Export | DXF**) and opening this layer in **AutoCAD** system these objects will be shown with the label assigned.

4.2.2.7. *Attaching vector objects to classifier*

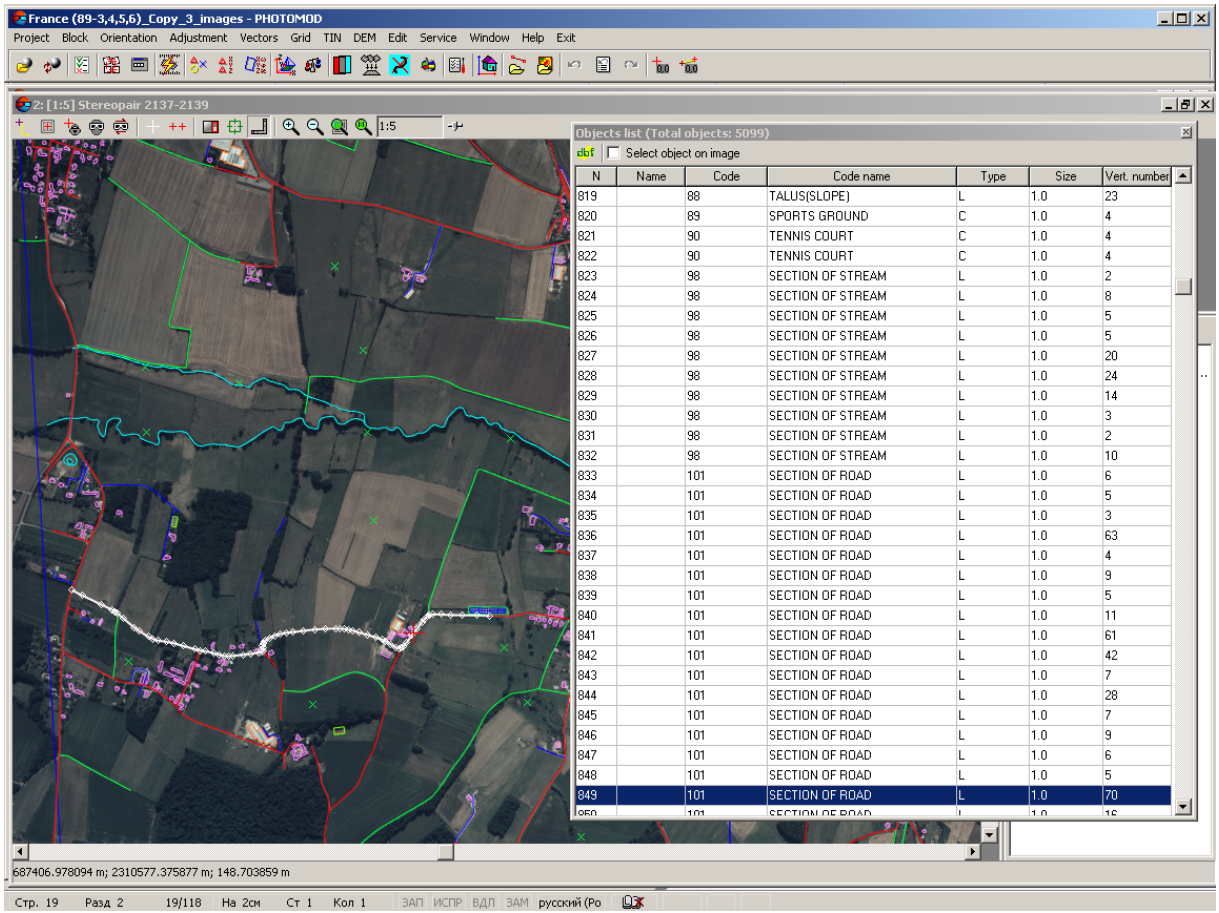
The classifier records may be attached to vector objects in two ways:

- **while the vector object creating**. Select the record you need in the **Classifer** window and start the vectorization. The code will be assigned automatically
- **attaching record to an existing object**. Select an object or a group of objects in 2D window and code to be assigned in the Classifier and click the icon (**assign code to selected objects**). See the chapter Objects selection.

See also the chapter 6 Creating vector objects and Changing object type.

4.2.2.8. *Object list*

The list of vector objects of the current project can be displayed in the **Object list** window (**Windows / Object list** menu).


France (89-3,4,5,6)_Copy_3_images - PHOTOMOD
Project Block Orientation Adjustment Vectors Grid TIN DEM Edit Service Window Help Exit
2: [1:5] Stereopair 2137-2139
Objects list (Total objects: 5099)
Select object on image
N Name Code Code name Type Size Vert. number
819 88 TALUS(SLOPE) L 1.0 23
820 89 SPORTS GROUND C 1.0 4
821 90 TENNIS COURT C 1.0 4
822 90 TENNIS COURT C 1.0 4
823 98 SECTION OF STREAM L 1.0 2
824 98 SECTION OF STREAM L 1.0 8
825 98 SECTION OF STREAM L 1.0 5
826 98 SECTION OF STREAM L 1.0 5
827 98 SECTION OF STREAM L 1.0 20
828 98 SECTION OF STREAM L 1.0 24
829 98 SECTION OF STREAM L 1.0 14
830 98 SECTION OF STREAM L 1.0 3
831 98 SECTION OF STREAM L 1.0 2
832 98 SECTION OF STREAM L 1.0 10
833 101 SECTION OF ROAD L 1.0 6
834 101 SECTION OF ROAD L 1.0 5
835 101 SECTION OF ROAD L 1.0 3
836 101 SECTION OF ROAD L 1.0 63
837 101 SECTION OF ROAD L 1.0 4
838 101 SECTION OF ROAD L 1.0 9
839 101 SECTION OF ROAD L 1.0 5
840 101 SECTION OF ROAD L 1.0 11
841 101 SECTION OF ROAD L 1.0 61
842 101 SECTION OF ROAD L 1.0 42
843 101 SECTION OF ROAD L 1.0 7
844 101 SECTION OF ROAD L 1.0 28
845 101 SECTION OF ROAD L 1.0 7
846 101 SECTION OF ROAD L 1.0 9
847 101 SECTION OF ROAD L 1.0 6
848 101 SECTION OF ROAD L 1.0 5
849 101 SECTION OF ROAD L 1.0 70
687406.978094 m; 2310577.375877 m; 148.703859 m

Selecting object in 2D window and in the list

Following objects are displayed in the **Object list** window: object numbers, codes, types, attributes. If **Select object on image** option is On the mouse click on the record in the object list selects the corresponding object on the image.

Object list can be saved to .dbf file by using icon in the upper part of the window

4.3. *Vector objects creation*

3D vector objects (points, lines, polygons and CAD-objects) are used for both vector maps and DTM creation. There is also a special automatically created vector object - pre-regions)

4.3.1. Vector object types

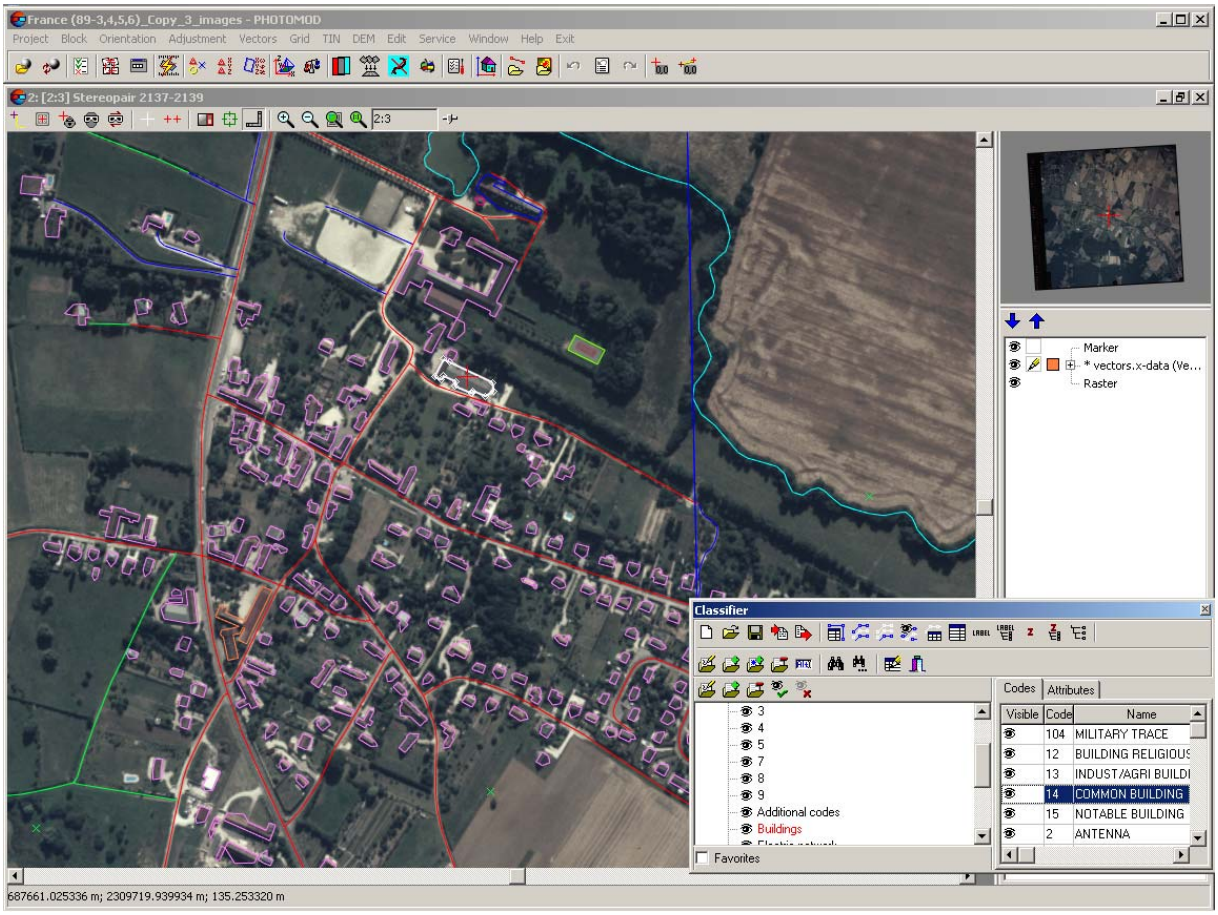
PHOTOMOD works with 3D vector objects of the following types:

- ***point*** – a point in space, defined by X, Y, Z coordinates
- ***line*** *(polyline)* – a set of XYZ points (***vertices***) connected by straight lines (***segments***). The polyline may be either closed or unclosed. In case of closed polyline its first and last vertices are spatially coinciding
- ***polygon*** – an area object. The polygon is closed automatically after finishing its creation. During polygon creation its first and last vertices are coincide spatially and are edited simultaneously.
- ***CAD objects*** – standard geometric figures, see the chapter Creating CAD-objects.

4.3.2. Vector objects creation

All marker related operations, described in the chapter Operations with marker are used during vector objects creation.
In order to start the vectorization process you should select an existing vector layer in the Layer manager window or create a new one (**Vectors | Create layer** or **Vectors | Create layer with classifier**)

Note: In order to increase the vectorization productivity, use special 3D or multibutton mice, hand wheels as well as programming of ordinary mouse buttons for different operations (see the chapter Mouse settings)



Creating vector objects

4.3.2.1. *Point creation*

- If the classifier is available select the point object you need in the classifier
- If you work with a layer without classifier activate point creation mode (by pressing **P** - key or using the icon (**point object creation**);
- locate the marker in 2D window to the position you need (see the chapter Operations with marker);
- press **Insert** to add a point.

4.3.2.2. *Polyline creation*

- If the classifier is available select the polyline-type code you need in the classifier
- If you work with a layer without classifier activate point creation mode (by pressing **L** - key or using the icon (**polyline object creation**);

- for each polyline vertex:
  -locate the marker in 2D window to the position you need (see the chapter Operations with marker);
  -press **Insert** key to add the vertex;
- press **Enter** or **Esc** to finish polyline creation.

Use the **Vectors | Topology | Close polyline** menu command (duplicated by the **Shift-C** keyboard shortcut) after the last vertex entering to close polyline in the process of its creation. Description of orthogonal polyline creating see chapter Orthogonal objects creation mode.

4.3.2.3. *Polygon creation*

- If the classifier is available select the polyline object you need in the classifier
- If you work with a layer without classifier activate point creation mode (by pressing **G** - key or using the icon (**polygon object creation**);
- for each polygon vertex:
  -locate the marker in 2D window to the position you need (see the chapter Operations with marker);
  -push **Insert** to add the vertex;
- press **Enter** or **Esc** to finish polygon creation.

See also the chapter Orthogonal objects creation mode about buildings vectorization.

4.3.2.4. *Creating **orthogonal** vector objects*

Vectorization of some objects (buildings, for instance) requires drawing line segments at the right angle. To turn on such **rectangle** mode use the following ways:

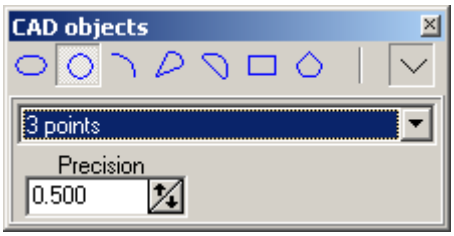
- Press the **A** key if you would like to draw some part of polyline in rectangle mode. Press and keep holding down the **A** key before adding next segment and release this key to continue the line in usual mode
- Push the icon of the main panel to turn on the constant “orthogonal” vectorization mode.

Rectangle mode is used for creating both polylines and polygons (see the chapters Polyline creation and Polygon creation).

If you need to create a closed orthogonal polyline, use **Vectors | Topology | Close polyline** command (**Shift-C** hot key) after entering last polyline vertex, and next press **Esc** key.

4.3.2.5. *CAD-objects creation*

In order to create vector objects of standard geometric shape, use a menu, opened by the command **Edit | Vectors create mode | CAD objects**.



CAD objects creation window

Push the button in this menu to open lower part of the window containing settings for each object type. The drop-down list is used for parameters selection, described below.

The following buttons are used to choose the type of the object to be created:

- Ellipse – if the button is pushed, you can start ellipse creating. Press **Insert**, then drag ellipse's semi-axis by mouse to needed size and click **Insert** once more to create ellipse.
- Circle – if the button is pushed, you can start circle creating. Press **Insert**. If the circle is build by two points, second press of **Insert** indicates the diameter of the circle. If the option **Center, radius** is selected, second press of **Insert** indicates the radius of the circle. If the circle is build by two points, press **Insert** three times to indicate three points, lying on the circle.
- Arc – if the button is pushed, you can start arc creating. Press **Insert**, and then move the marker to specify the arc's radius. Press **Insert** once more, then drag a polygon of needed size by mouse.
- Sector – if the button is pushed, you can start sector creating by pressing **Insert**. The second pressing of **Insert** specifies sector's radius, which could be both fixed and changeable, depending on the option selected in drop-down list. Then drag a polygon of needed size by mouse.
- Segment – if the button is pushed, you can start segment creating by pressing **Insert**. The second pressing of **Insert** specifies the beginning and the end of segment's arc or another parameter depending on the option selected in drop-down list. Then drag a polygon of needed size by mouse.
- Rectangle – if the button is pushed, you can start rectangle creating by pressing **Insert**. Then indicate three rectangle vertices by **Insert**, or another points of the rectangle depending on the option selected in the drop-down list. Then drag a polygon of needed size by mouse.
- Polygon – if the button is pushed, you can start polygon creating (regular convex figure) by pressing **Insert**. Then drag a polygon of needed size by mouse. Number of vertices should be input into appropriate field (3 – by default, possible values from 3 to 500).

Since broken lines approximate CAD-objects in PHOTOMOD system, you can change the object's curvature by changing the maximal distance from segment of broken line to curve between two closest vertices. This distance is called **precision** and could be adjusted for CAD-objects with curved segments (ellipse, circle, arc, sector, segment). Default value of this parameter is 0.5, in units defined by the user for the current project.

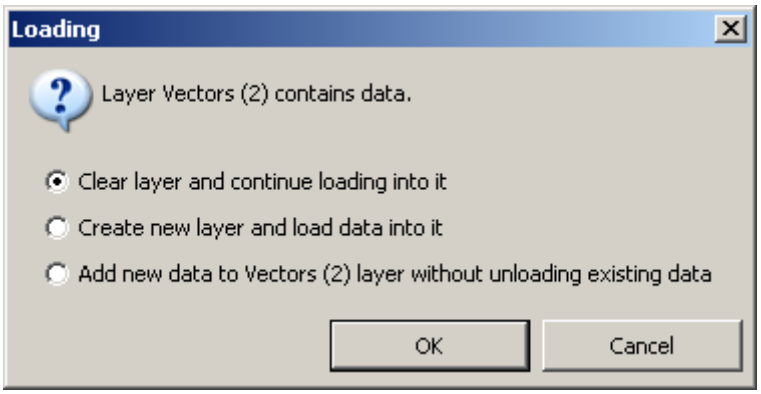
Note. Prior to creating of the CAD-objects (arcs) you should select the appropriate object type in the Classifer if the latter is used: (L) for linear CAD-objects (arcs), and (C) – for areal objects, see the chapter Classifier.

Once the object type and parameters are selected, start its creation in 2D window. To confirm the creation press **Enter** at the end. Or press **Esc** to cancel object creation.

You can also transform existing vector object(s) into geometric figures, as described in the chapter Transforming object to CAD-figure.

4.3.3. Loading vector data

Use **Vectors | Load** menu option to load existing vector objects to 2D window. This command opens a list of vector datasets available in the current profile (they have *.x-data - extension). If you select several vector datasets you can load them into one or several layers (by using the appropriate options). Corresponding vector layers appears in the Layer manager panel.
If there are some vectors already loaded you have several choices when loading new ones.


Loading
Layer Vectors (2) contains data.
Clear layer and continue loading into it
Create new layer and load data into it
Add new data to Vectors (2) layer without unloading existing data
OK
Cancel

- **Clear layer and continue loading into it** – vector data of the active layer are replaced with selected new ones.
- **Create new layer and load data into it** – vectors are added into a new layer.
- **Add new data to Vectors without unloading existing data** – vector data are added to the currently exiting vector objects of the active layer.

Vectors | Recent menu option is used for fast opening vector resources, you worked on last time.

4.3.4. Saving vector data

Vectors | Save and **Vectors | Save as** commands are use to save vector data to the source or another dataset correspondingly.
You can also go back to the last saved data by using **Vectors | Revert** command of the main system menu.

4.3.5. Switching between stereopairs

For vectorization of long objects continuing along the whole block of images (roads for example) it is convenient to digitize them going through all stereopairs without breaking. To pass to adjacent stereopairs use following commands:

- **Windows | Next stereopair** or main menu icon – allows to pass to the next stereopair in the strip (**Ctrl-Alt** + **right** arrow key)
- **Windows | Previous stereopair** or main menu icon – allows to pass to the previous stereopair in the strip (**Ctrl-Alt** + **left** arrow key)

Note that **PHOTOMOD** keeps zoom, parallax and marker position when switching between stereopairs for immediate and comfortable continuing the vectorization.

4.4. *Editing of vector objects*

4.4.1. Objects selection

Editing operations are applied to the selected vector object (group of objects). Objects may be selected manually (see the chapters Selection tools and Selection modes) or by a query in the classifier - by code, by layer etc. (see the chapter Classifier).
Use **Edit | Group selection** option of the main menu to access the vector selection tools.

4.4.1.1. *Selection tools*

Following tools are used to select vector objects:

- **double click** or **S** hot key to select a single polyline or polygon or **single mouse click** to select a single point/vertex (you should place the marker near the objects beforehand)
- **rectangle** (the icon of the main panel or menu command **Edit | Selection | Rectangle**). Allows to select vector objects inside the rectangle “drawn” by **mouse** along with pressed **Shift** key
- **polygon** (the icon of the main panel or menu command **Edit | Selection | Polygon**). Allows to select vector objects inside the polygon of an arbitrary shape. Press left mouse button and **Shift** key to start polygon drawing. As a result you have a first vertex and a rubber-line to the current cursor position. Use left mouse button for adding vertices, and finally double click left mouse button to finish objects selection by polygon.

4.4.1.2. *Selection modes*

You can set a selection mode using **Edit | Group selection** menu items or corresponding icons of the main panel:

- (**normal**) – selects objects and deselects currently selected objects
- (**add to selected**) – adds objects to a group. Each new selected object (or group of objects) is added to the current group of selected objects
- (**subtract from selected**) – cancels object (or group of objects) selection
- (**invert selected**) – invert operation. Selected objects are removed from a group if they are currently included and are added to a group if they are not currently included
- **Fully inside** - vector objects selection mode used to select objects which were fully inside in selection frame.
- **Partially inside** - This mode allows to select objects which are either entirely inside selection frame or intersect it, not necessarily having any points inside the frame.
- **At least one point inside** - vector objects selection mode used to select objects having at least one vertex inside the selection frame.

To select vector objects and their vertexes in consecutive order use **Vectors | Selection** menu command which opens the following submenu:

- Selection of previous object duplicated by the **Ctrl-<** keyboard shortcut or **Vectors | Selection | Select previous object** menu command.
- Selection of next object duplicated by the **Ctrl->** keyboard shortcut or **Vectors | Selection | Select next object** menu command.
- Select of previous polyline vertex duplicated by the (**<**) hot key or **Vectors | Selection | Select previous line vertex** menu command.

- Selection of next polyline vertex duplicated by the (**>**) hot key or **Vectors | Selection | Select next line vertex** menu command.

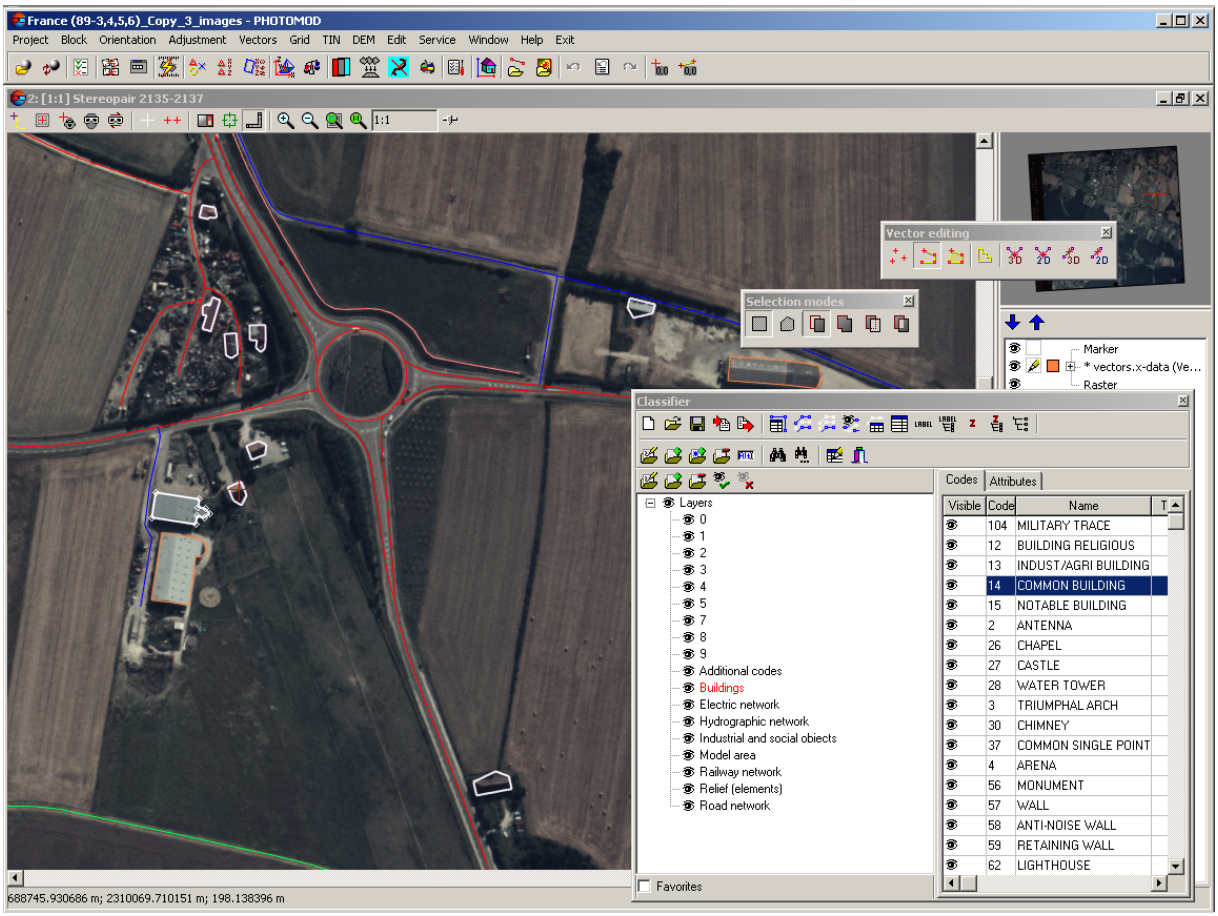
Use **Esc** key to cancel current selection.

4.4.1.3. *Selecting all objects in layer*

To select all objects of some layer, select a layer you need in the **Layer** list of **Classifier** window and click the icon (**select all group objects**). See also the chapter Classifier.

4.4.1.4. *Selecting objects of the same code*

To select all objects of the same code, select a code you need in Codes list of the **Classifier** window and click the icon (**select all objects with current code**). After that the objects with selected code will be highlighted in 2D window.
See also the chapter Classifier.



Selecting objects of the same code

4.4.2. Point editing

Once a point object is selected (by mouse click) you can:

- **delete it** using **Delete** key
- **move it** – press and hold down **Ctrl** key and move the point by **mouse** or **arrow keys** in XY plane and by **PgUp**, **PgDn** keys or mouse **wheel** by Z

- **move it to marker position** – use menu command **Vectors | Operations | Move point to marker** or **J** hot key. You can position marker manually (set the chapter Operations with marker) or by entering its coordinates in **Marker** window (see the chapter Marker window)

4.4.3. Polyline/polygon editing

4.4.3.1. *Vertex editing*

Polyline or polygon vertex is selected by the click of left mouse button. Following operations are available for selected vertex:

- **deleting - Delete** key
- **moving** – press and hold down **Ctrl** key and move the point by **mouse** or **arrow keys** in XY plane and by **PgUp**, **PgDn** keys or mouse **wheel** by Z
- **moving it to marker position** – use menu command **Vectors | Operations | Move point to marker** or **J** hot key. You can position marker manually (set the chapter Operations with marker) or by entering its coordinates in **Marker** window (see the chapter Marker window)
- **line orthogonalization on the vertex** – menu command **Vectors | Operations | Orthogonalization forward** (duplicated by **Ctrl-F**) and **Vectors | Operations | Orthogonalization backward** (duplicated by **Ctrl-B**) causes moving the next (preceding) vertex in such a way that the angle at the selected vertex becomes equal to 90° or 180°. See also the chapter Creating orthogonal objects

Note: you can select a single vertex of polyline object vertex for editing (without selection of the object itself)

Note: if the Joint points edit mode is enabled, linked common vertices are moved together.

Note: if the Joint points edit mode option is enabled and the some number of vertices of different objects is already has "linked", then for their following simultaneous editing is necessary to use snapping to vertices (see chapter Types of snapping), and the Vectors | Operations | Move point to marker menu command.

4.4.3.2. *Vertex adding*

To insert a vertex to the selected polyline / polygon place the marker to the position you need and press **Insert**. If you select a vertex of polyline by **Alt-S** hot keys the **Insert** operation inserts a new vertex *after* the selected one.

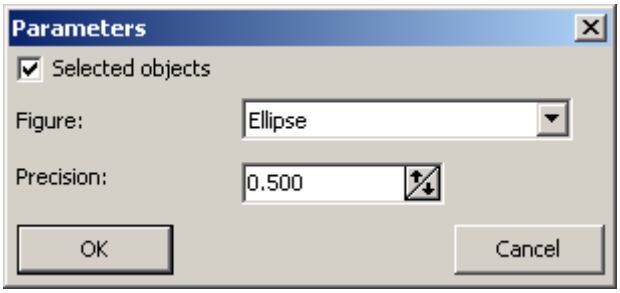
Note: if the Joint points edit mode option is enabled then inserting a vertex in the common segment of several polylines between two linked vertexes, inserts it into all of them.

4.4.3.3. *Polyline continuing*

To continue a selected polyline, make “active” its first or last vertex (**Alt-S** hot keys) and continue vectorization process in a standard way.

4.4.3.4. *Converting objects to figures*

To convert selected or all objects of the current layer into geometric figure use menu command **Vectors | Operations | Convert to figure**:



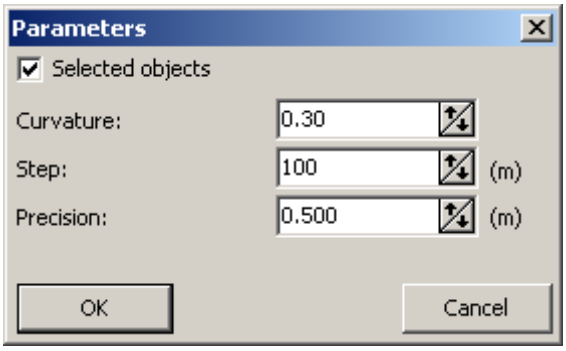
Conversion parameters

In the window appeared select figure type (ellipse, circle, rectangle, square) and setup conversion precision in appropriate fields. Vector objects presented in the curves form are approximated by polylines in the PHOTOMOD system.
The approximation accuracy that is the maximum distance from the polyline segment up to the curve at the section between the two nearest vertices, is set by user. By default it is equal to 0.5 in units of the current project (usually in meters).

4.4.3.5. *Interpolating polyline*

Polyline (polygon) interpolation (**Vectors | Operation | Interpolate**) is used to smooth polylines by adding vertices along calculated invisible Bezier curve. The typical application of smoothing is vectorization of road junctions. Interpolation parameters are as follows:

- **Curvature** – the degree of the curvature (recommended range of values **0.1 – 1.0**)
- **Step** – is a distance between vertices in output smoothed polyline
- **Precision** – defines whether to add vertices into some fragments of the output polyline. If the distance between the Bezier curve and output polyline is greater than **Precision** value, there will be more vertices added unless the criteria is satisfied.



Interpolate operation may by applied to all objects or to selected ones by using the appropriate option.

4.4.3.6. *Interpolation of polyline elevations*

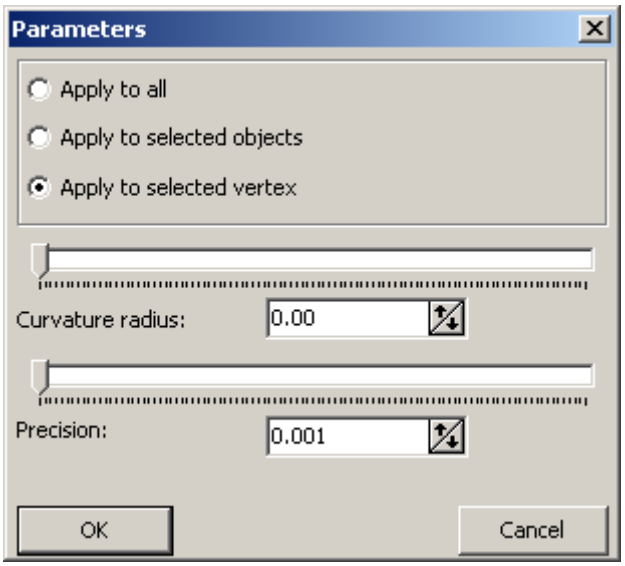
During lines vectorization on monotonous relief you can interpolate Z-values of polyline vertices. For that press **Z** key or select the command **Vectors | Operations | Interpolate elevations**) when creating polyline. At that heights of vertices located between first and last

vertices are interpolated proportionally to the distance between intermediate vertices and depending on the heights difference between first and last vertices.

You can also interpolate elevations of vertices in existing line if it is selected in 2D window.

4.4.3.7. *Rounding off corners*

To round off (smooth) the corners at vertices of vector objects (polylines and polygons) use menu command **Vectors | Operations | Round off corners.**



Smoothing parameters

Process parameters are as following:

- **Curvature radius** – the less the value the closer smoothing curve to the vertex of the initial broken line.
- **Precision** – you can change the object corner curvature by changing the maximal distance from segment of source broken line to curve between two closest vertices. This distance is called **precision.** Default value of this parameter is 0.5, in units defined for the current project (usually meters).

The operation can be applied to all objects, selected object and selected vertex in accordance with corresponding option.

4.4.3.8. *Moving polyline*

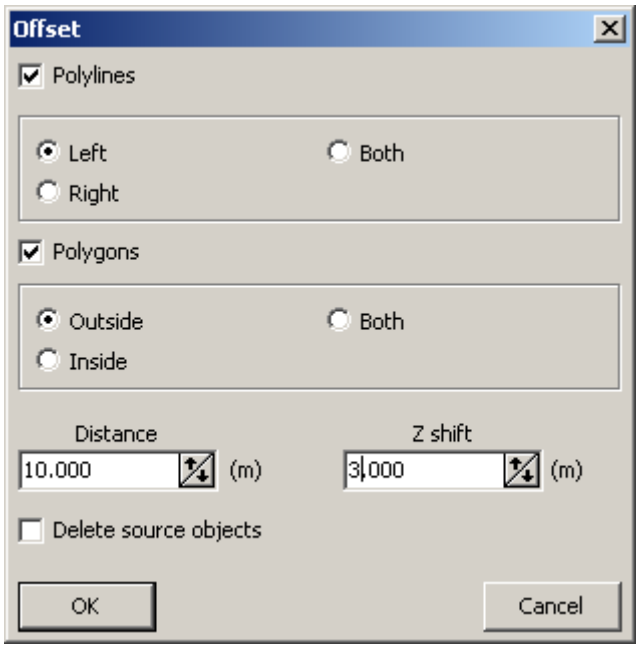
In order to move a polyline select it (e.g. by mouse double click), and move it by mouse along with held down **Ctrl** key and left mouse button, or with arrow keys and held down **Ctrl** key. To move polyline along Z use **Ctrl-PgUp**, **PgDn** hot keys or mouse wheel rotation with **Ctrl** key held down. There should be no selected vertices in the polyline.

4.4.3.9. *Deleting polyline*

Use **Delete** key to delete a selected polyline. If the polyline contains a selected vertex, first pressing the **Delete** key will delete the vertex and the second pressing will delete the whole polyline.

4.4.3.10. *Building buffer zone*

The buffer zone building operation (menu command **Vectors | Operations | Buffer zone**) creates a polyline or polygon, which is parallel to the source vector object (selected in 2D window) and is located at the selected distance from it.



Buffer zone creation parameters

Buffer zone can be created both for line and area objects. If the object of polyline type is selected in 2D window then in the window opened the **Polylines** check box is marked and you can select an option of location of buffer zone relatively to the object selected:

- **Left** – the polyline is created “leftward” from the source one correspondingly to its vertices numbering
- **Right** – the polyline is created “rightward” from the source one correspondingly to its vertices numbering
- **Both** – two polylines are created on both sides of the object selected

If the object of polygon type is selected in 2D window, then in settings window the appropriate check box is marked and you can select one of the following options:

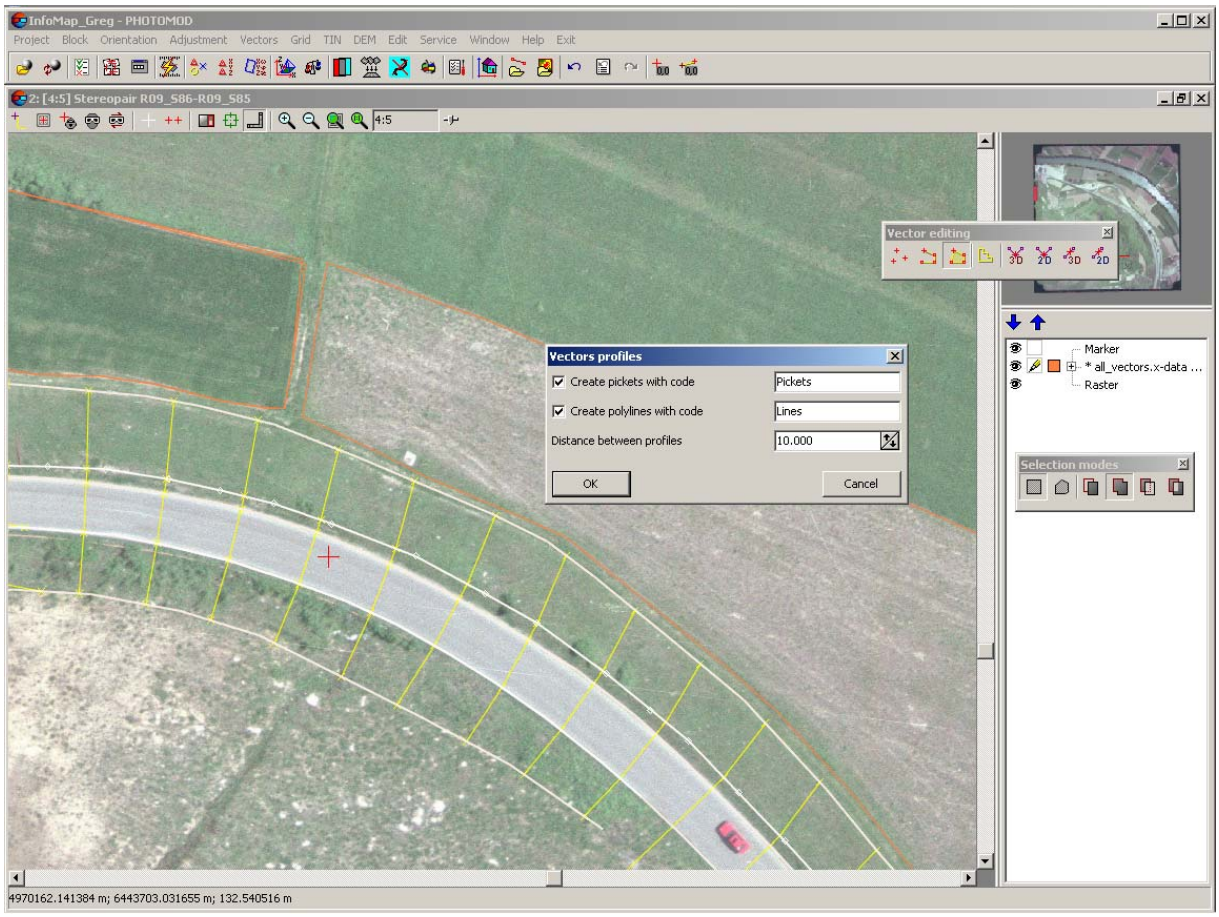
- **Outside** – outside of initial polygon
- **Inside** – inside of initial polygon
- **Both** – two polygons are created on both sides of the object selected.

Distance – distance in meters from selected polyline (buffer zone width)
Z offset – moving polyline vertices at selected offset by height

When buffer zone is created the initial object could be deleted if the appropriate option is selected.

4.4.3.11. *Creating profiles through vector objects*

You can create profile lines perpendicular to the group of linear objects. This is a good way to describe for instance, the dam, which “consists of” two lines at both feet and edges. In case of using layer with classifier the resulting profiles (linear or point vector objects) are attached to the selected codes in Classifier and can be used for various applications.

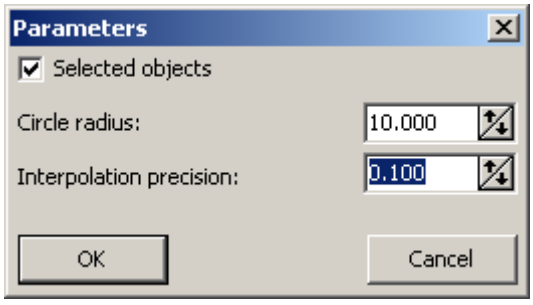


Profiles through vector objects

Prior to create profiles select vector objects in “add to selection” mode (for example, road edges), then select one of them (it will be shown in bold) and call menu command **Vectors | Operations | Vectors profiles**. In the window opened set the step between profile lines (in meters), which will be created at right angle to highlighted vector object (in the **Distance between profiles** field). You may save profile lines and/or intersection points of profiles with selected objects as polylines and pickets codes, by selecting the options **Create pickets with code** or **Create polylines with code** and input code names. At that the pickets and polylines will be attached to Classifier table as codes of linear (L) and point (P) objects, and placed to the its root directory.

4.4.3.12. *Circles around points*

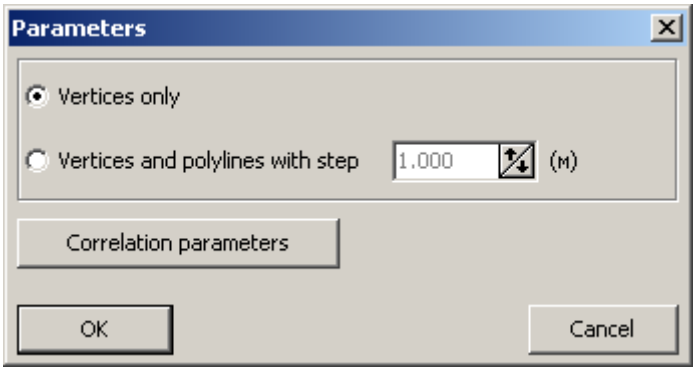
This feature is designed to create circles with the given radius around vertices of vector objects. To use this feature, required objects should be selected (or not, if it is necessary to process all objects) and then the **Vectors | Operations | Circle around points** menu command is called, which opens window with the following parameters:



The **Selected objects** option allows processing all or selected objects.
The **Circle radius** option specifies the required circles radius in project coordinate units.
The **Interpolation precision** option sets the maximum difference between segments of broken line which approximates the circle, and the circle itself.

4.4.3.13. *Project on stereomodel*

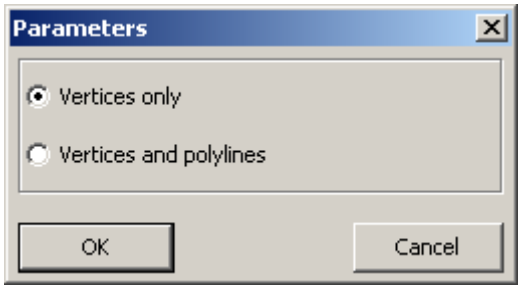
This feature allows automatically projecting vector objects on relief of active stereopair. To use this feature, one should open stereo pair, load vector objects and use the **Vectors | Operations | Project on stereomodel** menu command which opens window with the following parameters:



There are two projection modes:
• The **Vertices only** mode - in this case the vector objects vertices and point objects are projected.
• The **Vertices and polylines with step** mode - in this case, along with the vertices projection is performed by splitting linear objects into segments with a given step and projecting resulting intermediate vertices.
The **Correlation parameters** button calls a standard window of correlator parameters, see chapter Correlator parameters.

4.4.3.14. *Projection on TIN*

This feature allows to automatically project the loaded vectors on the loaded TIN.
To use this feature, one should load vectors, TIN and use **Vectors | Operations | Project on TIN** menu command which opens window with the following parameters:



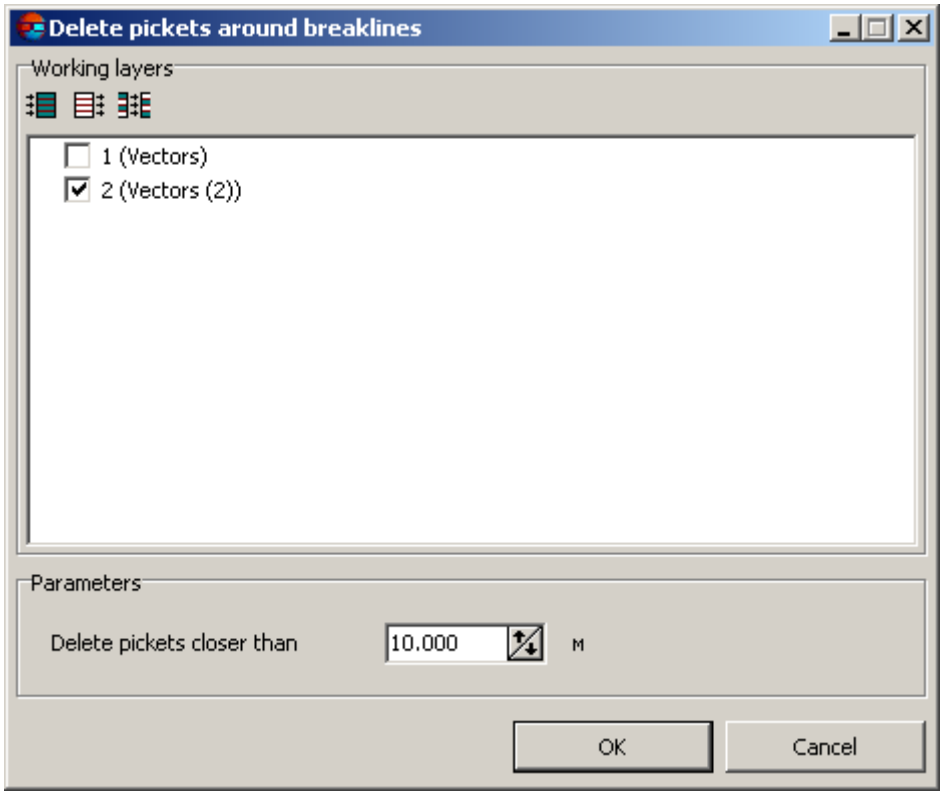
There are two projection modes:
• The **Vertices only** mode - in this case, the vector objects vertices and point objects are project on TIN.
• The **Vertices and polylines** mode - in this case, along with projection of vertices, line objects are intersected with TIN triangle edges and the resulting intermediate vertexes are also projected.

4.4.3.15. *Quick interpolate*

The **Quick interpolate** operation is designed for quick smoothing corners of linear vector objects and can be applied in cases where the accuracy and quality of the result does not really matter. The operation consists in replacing the vertex of each polyline by he middles of polyline edges. To perform the operation use the **Vectors | Operations | Quick interpolate** menu command. It will be prompted to choose whether to perform interpolation for all linear objects or only for selected ones.

4.4.3.16. *Deleting pickets around line objects*

The **Delete pickets around line objects** operation is designed to "cleanse" of pickets which are designed for TIN building, in case the pickets which are located close to the linear objects, are in conflict with these linear objects. To perform the operation use the **Vectors | Operations | Delete pickets around line objects** menu command which opens window with the following parameters:



If the TIN layer is active, then the basic vector layers are selected by default in the **Delete pickets around breaklines** window. If the vector layer is active, then it is selected by default. Otherwise, all layers are selected by default
All the layers selected are processed simultaneously, i.e. for each linear object of each layer the pickets which located closer then specified distance are removed from all the processed layers.

4.4.3.17. *Converting polygons to points*

The **Convert polygons to points** operation is designed to convert all or selected closed polylines (polygons) to point objects. For example, the operation can be used to convert settlements represented by polygons to point objects.

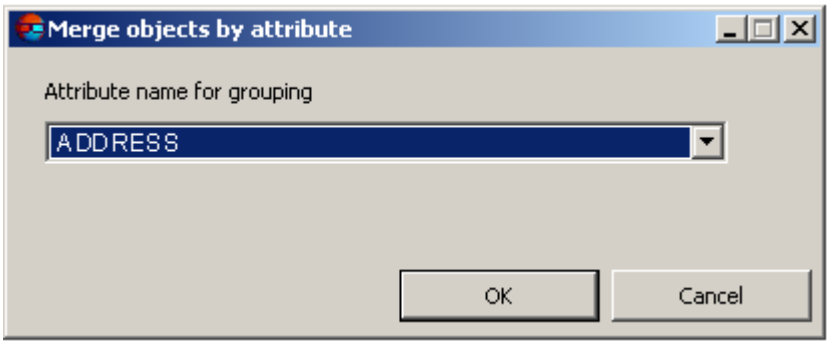
To perform the operation use the **Vectors | Operations | Convert polygons to points** menu command. If the operation is performed on the layer with classifier, then code type P (point) should be selected in the classifier, otherwise error message will be displayed.
If there are selected objects in layer, then user is invited to select if all objects or only selected ones should be changed.
After running this command, closed polylines (all or selected) are converted to point objects which are placed at the mass center of points constituting the boundary of source objects. Selected code in the classifier is assigned to new point objects (in the case of the layer with classifier), as well as all the attributes of the source linear objects.
Unclosed linear objects as well as point ones in the result of running command are not modified.

4.4.3.18. *Merging point objects by attribute*

The **Merge point objects by attribute** operation is designed to find all groups of point objects with identical values of a given attribute, and to convert each group of point objects into point object which is located at the mass center of the source objects group. For example, this operation can be used to generalize the map on which various real objects are marked by groups of point objects, and object attributes of each group contain the name of a real object, which these point objects are related to.

Note: If it is necessary to merge groups of closed polygons following the same principle, it is possible to first use the Convert polygons to points command, see the chapter Converting polygons to points.

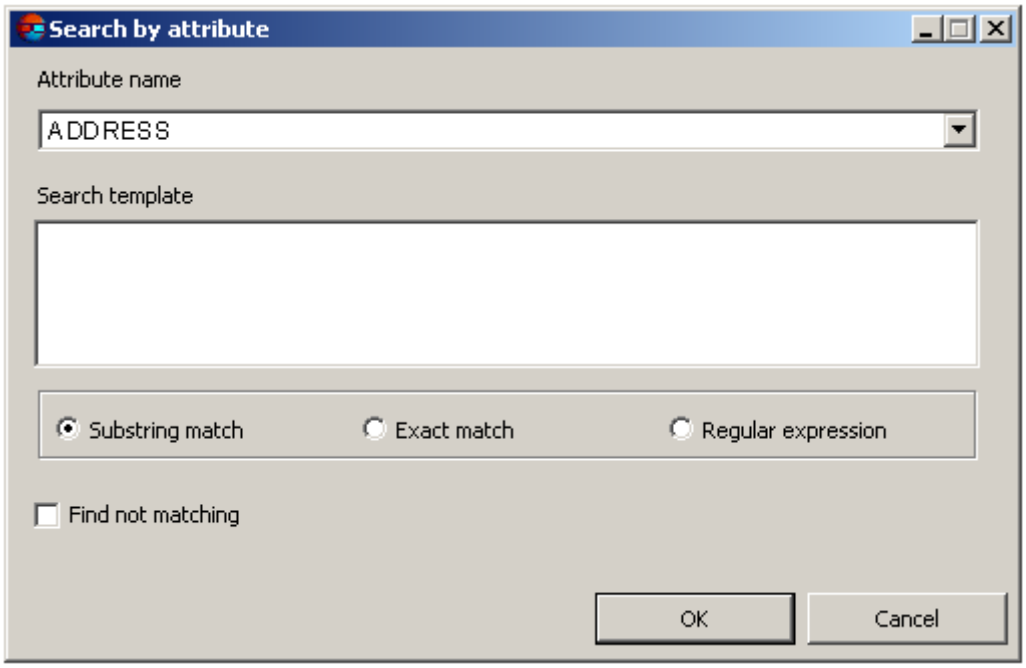
Use the **Vectors | Operations | Merge point objects by attribute** menu command to perform the operation. The dialog window to select the attribute name, by which values will be merged, opens before the operation.



The appropriate message is displayed if there is no point object which has an attribute with the specified name in the current vector layer. Otherwise, search starts for all point objects with the specified attribute; they are grouped by the value of this attribute, then in each group all objects except one (chosen randomly) are deleted, and this remainder object is moved to the mass center of the source group.

4.4.3.19. *Searching objects by attribute value*

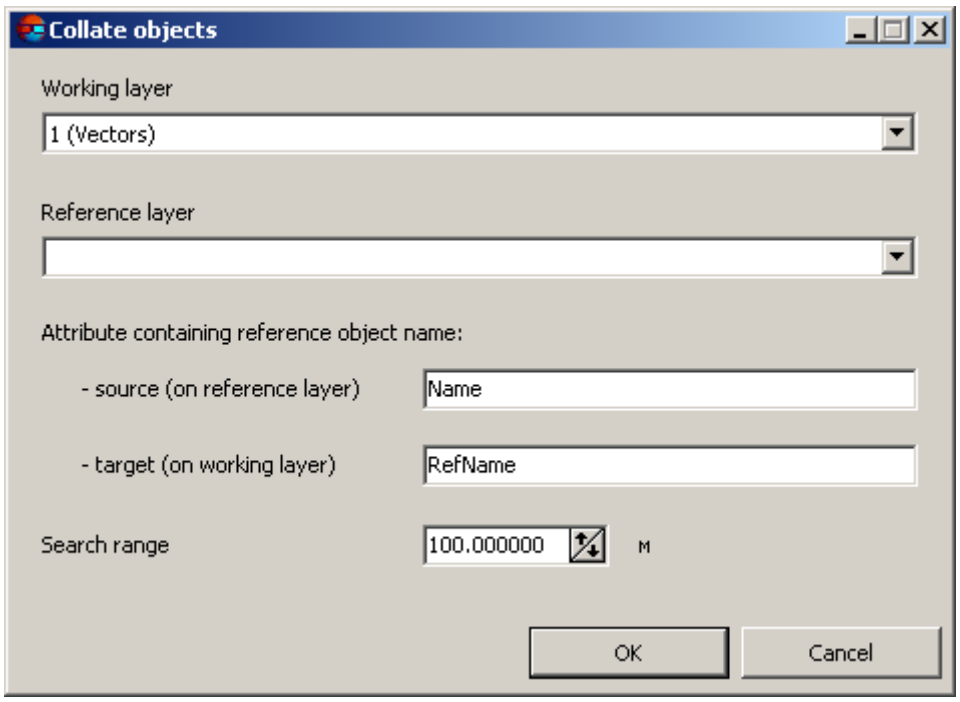
Searching objects by attribute value is performed using the **Vectors | Operations | Select by attribute** menu command. The dialogue window with search parameters is displayed on this command:



Attribute name field sets the attribute by which search will be performed.
Search template field allows user to enter text to search for. The value which is entered in this field is interpreted according to the specified search mode:
• **Substring match**. The object is satisfying to the condition, if the textual representation of the attribute value contains the template string which is used for case-insensitive comparison.
• **Exact match**. The object is satisfying the condition, if the textual representation of the attribute value is exactly the search template, case sensitive.
• **Regular expression**. The object is satisfying the condition, if the textual representation of the attribute value matches the regular expression entered in Search template field.
If **Find not matching** checkbox is set, only the objects which do not have the attribute or its value does not satisfy the search condition are shown.
Objects found this way (satisfying or not satisfying the search condition) are selected taking into account the current group selection mode (see chapter Selection modes).

4.4.3.20. *Collating objects*

Collating objects operation is designed for searching objects at the same vector layer, closest to objects of other vector layer. This operation is quite special and, for example, can be used to automate analysis of differences between two variants of file with elements of exterior orientation, if there is a probability that in one of them names of images and coordinates of projection centers are incorrectly matched.
Use **Vectors | Operations | Collate objects** menu command to perform the operation.
The dialogue window is displayed with the following parameters:



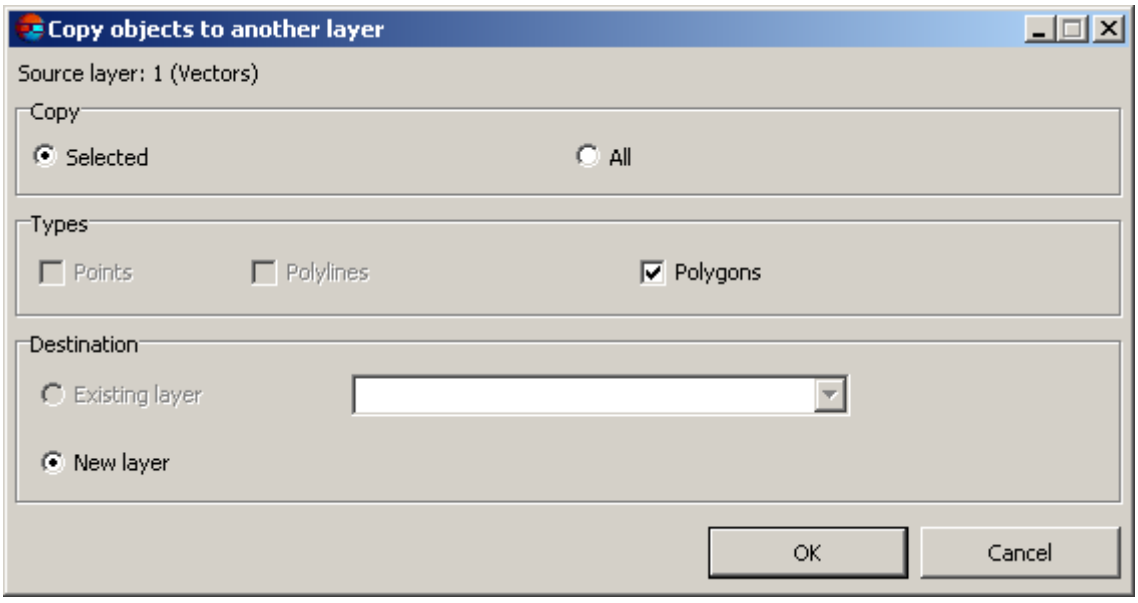
The performed operations are defined by parameters specified in this window, as follows:

- For each object of layer, set by **Working layer** field, the closest object of the layer, set in the **Reference layer** field, is matched, and the search is performed in the area with radius which is specified in **Search range** field.
- If an object of reference layer is found containing a string attribute with object's name (**Attribute containing reference object name** field is **source**), then the value of this attribute is saved into a given attribute of an object of working layer (**Attribute containing reference object name** field is **target**).

As a result of performing operation, every object of working layer contains attribute with a name of a closest object of reference layer; for further analysis a working layer can be exported into a format supporting tabular representation of attributes (for example, as DBF file - see chapter Import-Export) to create a list of correspondence between objects of one layer and objects of another.

4.4.3.21. *Copying to layer*

Use **Vectors | Operations | Copy to layer** menu command to copy vector objects from one layer to another. The dialogue window with copying parameters is displayed on this command:



Copy option allows user to copy **All** or **Selected** vector objects.
Types option allows user to select objects to copy depending on their type.
Destination option specifies the layer to which the vector objects will be copied. This may be an **Existing layer** or a **New layer**.

4.4.3.22. *Copying/pasting using clipboard*

Selected vector objects can be copied to clipboard by pressing **Ctrl-C** keyboard shortcut or **Vectors | Clipboard | Copy** menu command. Vector objects from clipboard may be pasted by pressing **Ctrl-V** keyboard shortcut or **Vectors | Clipboard | Paste** menu command. Thee **Vectors | Clipboard | Cut** menu command or **Ctrl-X** keyboard shortcut are used to move the object, for example, to the other layer (with cut and consecutive paste operation). **Vectors | Clipboard | Paste into marker position** menu command or **Ctrl-Shift-V** keyboard shortcut is designed to move object, for example, within the bounds of one layer. Marker position is used as a reference point while pasting the vector objects. Thus, in the case if marker is not moved between the copying and pasting operations the objects will be pasted in the same coordinates as the source; otherwise they will be shifted by the vector, which connects the marker positions at the moments of copying and pasting.

4.4.3.23. *Operations with group of objects*

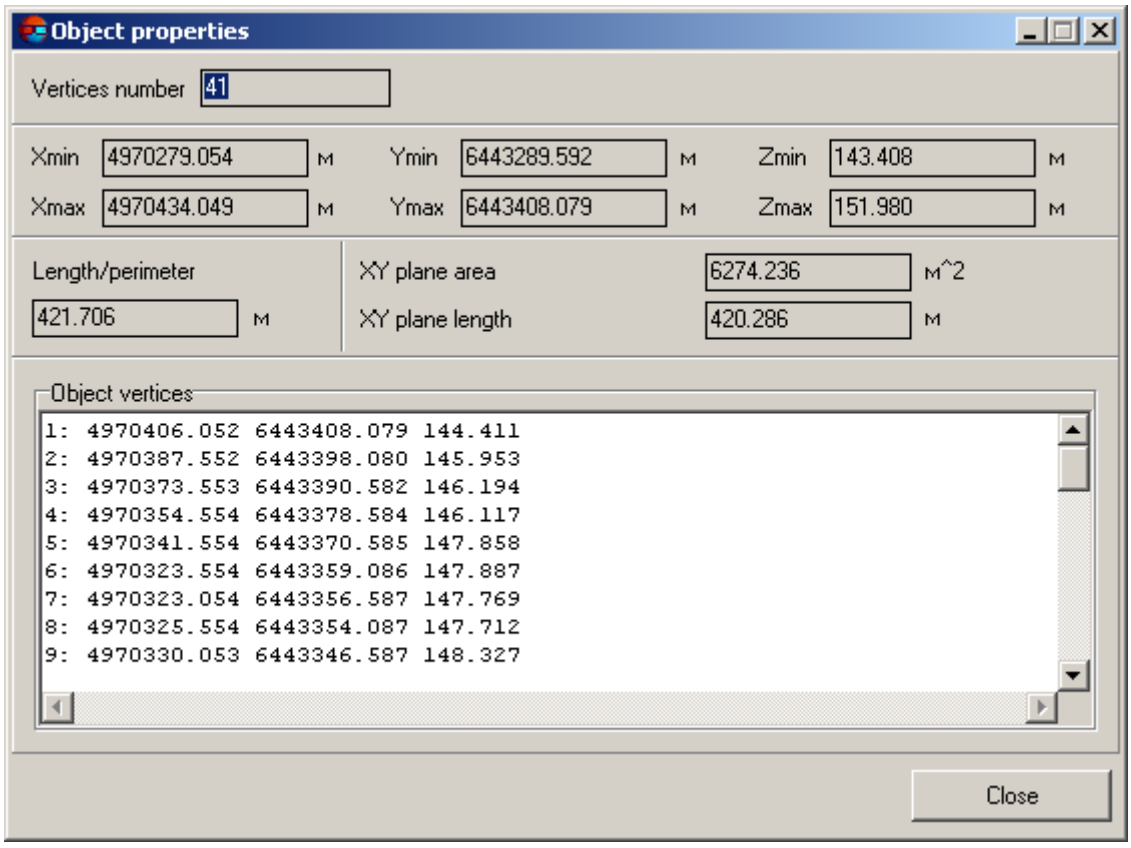
Following operations with a group of selected objects (see chapter Objects selection) are available:

- **Deleting** (**Delete** key)
- **Moving** – press **Ctrl** key and move objects in XY-plane by mouse or arrow-keys and along Z-axis by **Ctrl**-**PgUp**, **PgDn** keys or the mouse wheel with Ctrl pressed.
- **Converting object type** (see chapter Object type conversion)
- **Building buffer zone** (see chapter Buffer zone)
- **Interpolating** (see chapter Polyline interpolation)
- **Profiling** (see chapter Building profile through selected objects).
- **Building circles around points** (see chapter Circle around points).
- **Projection on stereomodel** (see chapter Project on stereomodel).
- **Quick interpolate** (see chapter Quick interpolate).
- **Converting polygons to points** (see chapter Converting polygons to points).
- **Deleting pickets around line objects** (see chapter Deleting pickets around line objects).
- **Merging point objects by attribute** (see chapter Merging point objects by attribute).

- **Selection by attribute** (see chapter Searching objects by attribute value).
- **Collating objects** (see chapter Collating objects).

In addition, it is possible to look at some properties of an object selected in **2D** window using the **Object properties** window, which is displayed by **Vectors | Line object properties** menu command (duplicated by **I** hot key):

- The number of vertices
- The maximum and minimum object coordinates
- Length/perimeter
- Area in plane (in case of closed polygon)
- Length in plane.



In bottom window there is a table with X, Y and Z coordinates of the object vertices, numbered according to their vectorization order.

4.4.3.24. *Object type conversion*

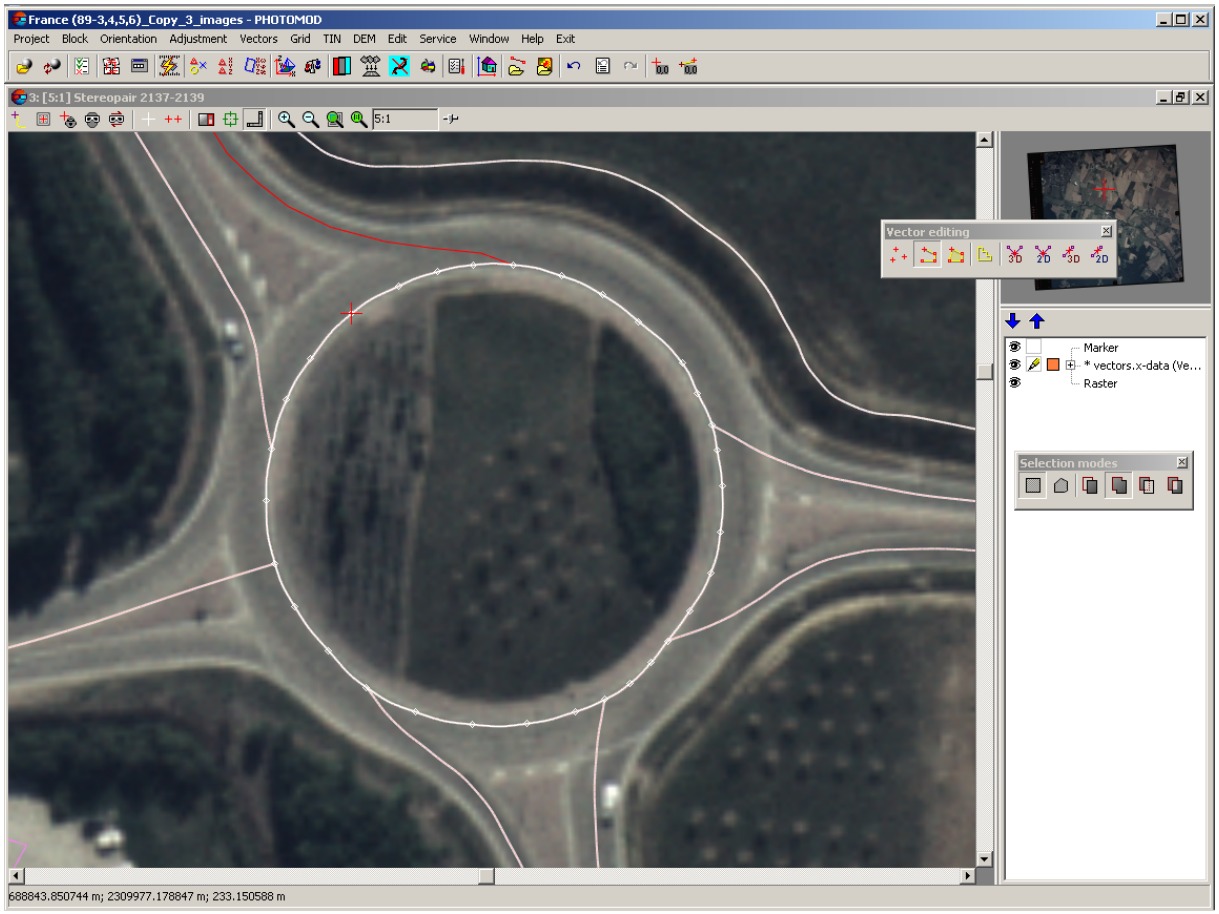
You can convert vector polygons to vector polylines and vice versa.
Select a polyline or a group of polylines (see the chapter Objects selection), select in the **Classifer** window a polygon code and click the icon (**assign code to selected objects**). As a result the polylines get closed and their code is changed.
The opposite operation converts polygons to polylines. In this case you have to select polyline code in the **Classifier** window

4.4.4. Curves editing

4.4.4.1. *Curves creating mode*

Curves creation mode is toggled by using **Edit | Vectors create mode | Curves** option (duplicated by icon of **Vectors** toolbar)
For the layer with classifier you should:

- select an appropriate polyline or polygon code (**L** or **C** type)
- add curve vertices one by one by **Insert** key
- press **Enter** to finish the curve creation.


France (89-3,4,5,6)_Copy_3_images - PHOTOMOD
Project Block Orientation Adjustment Vectors Grid TIN DEM Edit Service Window Help Exit
3: [5:1] Stereopair 2137-2139
Vector editing
Marker
* vectors.x-data (Ve...
Raster
Selection modes
688843.850744 m; 2309977.178847 m; 233.150588 m

When working without classifier you should select a polyline or polygon object type in **Vectors editing** toolbar.

4.4.4.2. *Polylines to curves conversion*

Use **Edit | Curve transforms | Convert polylines to curves** command to convert selected polylines to curves (duplicated by icon).

4.4.4.3. *Curves to polylines conversion*

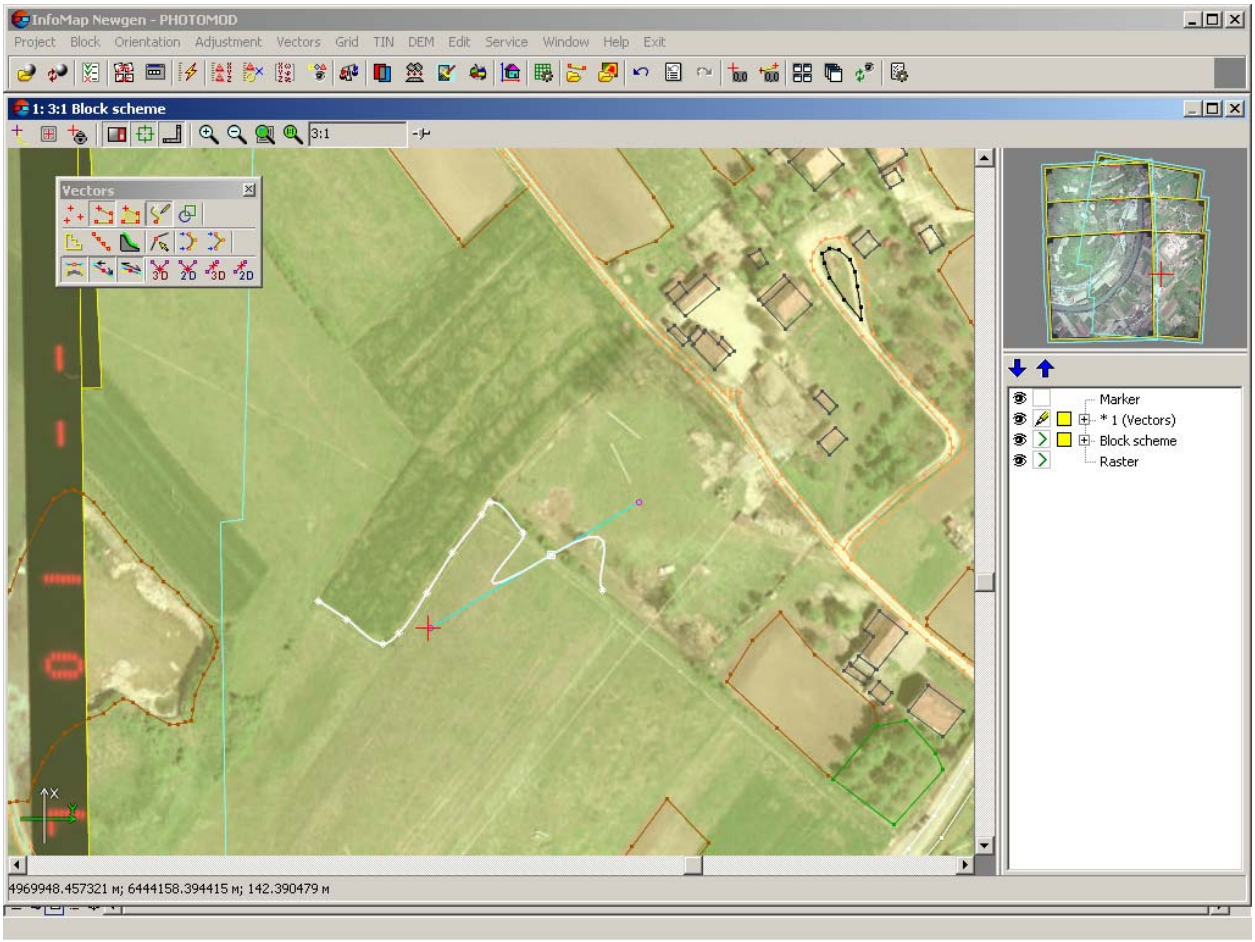
Use **Edit | Curve transforms | Convert curves to polylines** command to convert selected curves to polylines (duplicated by icon).

4.4.4.4. *Automatic smoothing adjustment*

PHOTOMOD allows you to adjust smoothing level for each added curve vertex automatically (**Edit | Curve transforms | Resmooth on editing** option (duplicated by icon)). Any operation of the curve editing in this case keeps the smoothing level equal to the value entered in **Service | Settings | Vectors | Default curve smoothing** text field.

4.4.4.5. *Curve segments editing*

To start editing the segments of curve, use the **Edit | Curve transforms | Edit control nodes** menu command (the mode is activated by pushing the **X** hot key or toolbar button). Using the **Ctrl** key and left **mouse** button user can edit the position of segments of the smooth curves created, «pulling» the **curve nodes**.

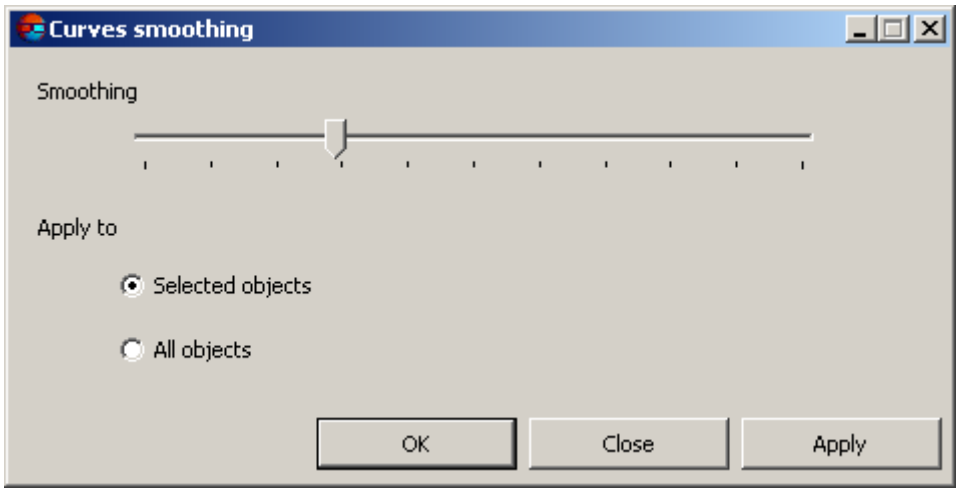


Control markers (in the figure in blue)

The **Smooth editing of curve control nodes** option (duplicated by pushing the toolbar button) turns on/off the possibility of the angles formation during editing segments of curve.

4.4.4.6. *Smoothing control*

Method of controlling the smoothing is activated by the **Edit | Curve transforms | Curves smoothing** menu command. The degree of smoothing is determined using the like-named slider. The operation of smoothing can be applied to all objects and selected objects by selecting the appropriate radio button.



The degree of curves smoothing

4.4.5. Topology

Topological operations are based on the following principle: the vertices which were marked as "linked" (**Vectors | Topology | Make vertices of selected objects linked** menu command) and located closer to one another than the specified distance (see chapter Settings of vector objects displaying) are considered to be common vertex and edited simultaneously. *Linked vertices* (topologically united) which are moved and deleted as one, in the case if the **Joint points edit mode** option is turn on; are created using **Vectors | Topology** menu commands*:*

- **Make vertices of selected objects linked**
- **Connect to point** (see chapter Connection to point)
- **Connect to polyline** (see chapter Connection to polyline)
- **Continue along polyline** (see chapter Continuing along polyline)
- **Auto-close along polyline** (see chapter Closing along polyline)

4.4.5.1. *Segment deleting*

Delete segment operation deletes a segment between two polyline vertices. As a result two polylines are created instead of unclosed polyline. Segment deleting will unclose closed polyline.
Select a polyline, place the marker near the segment you want to delete and select **Vectors | Topology | Delete segment** command or press **Shift-D** hot keys.
When deleting the segment in polygon object, the object will be transformed into polyline. This operation is available only for layers with classifier. Before deleting the segment from polygon, you should select polyline code in **Classifier** window.

4.4.5.2. *Merging polylines*

To merge two unclosed polylines, select them (see the chapter Object selection) and use menu command **Vectors | Topology | Merge two polylines** or hot keys **Shift-P.** As a result the segment will connect two closest first or last vertices of polylines. In case the source lines have different codes attached the output line will get the code currently selected in the **Classifier** window (see the chapter Classifier)

4.4.5.3. *Merging polygons*

To merge two overlapping or having common border polygons, select them (see the chapter Object selection) and use menu command **Vectors | Operations | Polygons merging** or hot keys **Shift-G.** In case if the source polygons have different codes attached the output polygon will get the code currently selected in the **Classifier** window (see the chapter Classifier), otherwise the output polygon retains the code of source polygons.

4.4.5.4. *Closing polyline*

In order to close polyline you should select it and press **Shift-C** or use a command from the main menu: **Vectors | Topology | Close polyline**. In case of layer with classifier, a polygon code (type **C**) must be selected beforehand in the **Classifier** window.

4.4.5.5. *Unclosing polyline*

Use **Vectors | Topology | Unclose polyline** menu item or hot keys **Shift-B** to unclose a selected polyline. As result the segment connecting the first and last vertices will be deleted. In case of layer with classifier you should select a polyline code beforehand in the **Classifier** window.

4.4.5.6. *Splitting polyline*

To split polyline into two polylines, place the marker to the needed break position and select menu command **Vectors | Topology | Split polyline** (or hot keys **Shift-X**). You can simply split polygon object in non-classifier layer. In case of layer with classifier you should select a polyline (type **C**) code in the **Classifier** window beforehand.

4.4.5.7. *Connection to point*

To connect the polyline being created to the existing one at a vertex, use the **Vectors | Topology | Connect to point** menu command (duplicated by the **Shift-V** hot keys). As a result both lines have common vertex and editing of it causes simultaneous modification of both objects (see chapter Topology).

4.4.5.8. *Connection to polyline*

To connect the polyline being created to the existing one in arbitrary point of its segment, use the **Vectors | Topology | Connect to polyline** menu command (duplicated by the **Shift-L** hot keys). The new vertex is created on the intersection of existing line segment and “rubber” continuation of new line. As a result both lines have common vertex and editing of it causes simultaneous modification of both objects (see chapter Topology).

4.4.5.9. *Continuing along polyline*

The operation is especially important for vectorization of objects, boundaries of which should coincide precisely, land parcels for instance.

When you **draw a polyline along the existing one** you can start and complete the common part of new and existing polylines using the following two ways.

How to create the first common vertex of two linear objects

• *In a vertex of existing line*
If there is existing polyline along which new object should be drawn start new linear object vectorization. Place the marker near the vertex of the existing object and press **Shift-V** hot keys, after that the vertex becomes the first common point of both lines.

• *In arbitrary point of existing line*
If there is existing polyline along which new object should be drawn start new linear object vectorization. Place the marker near arbitrary point (not vertex) of the existing line and press **Shift-L** hot keys (or select the command **Edit | Topology | Connect to line**), after that this point becomes a vertex and the first common point of both lines.

How to create the last common vertex of two linear objects

• *To selected vertex of existing line*
Place the marker near the vertex supposed to be the last common point for both lines and press **Shift-S** hot keys to select this vertex. Then use **Shift-F** hot keys (or select menu command **Edit | Topology | Continue along polyline | To selected vertex**). After that the existing and newly created lines will coincide topologically on the selected part. At that both lines are editable independently. In this case the vertexes of both objects can be edited separately and simultaneously, depending on the **Joint points edit mode option** (see chapter Topology).

• *To arbitrary point of existing line*
Place the marker to an arbitrary point near existing polyline (near the point supposed to be the last common vertex of both lines) and press **Shift-A** hot keys (or select the command **Edit | Topology | Continue along polyline | To marker position**). The last common vertex of two lines will be created in cross-point of existing line and its projection on it. If the marker is placed so that there is no it's projection on any polyline segments, the additional segment will be created on the line and new polyline will be continued to the closest vertex.

After creating the last common vertex of two linear objects continue new line creation, and press Enter to complete it.

As a result two polylines have common part bounded by two consequently selected vertices. When editing the vertex belonging to several objects, all these objects are changed simultaneously, in the case if **Joint points edit mode option** is turn on (see chapter Topology).

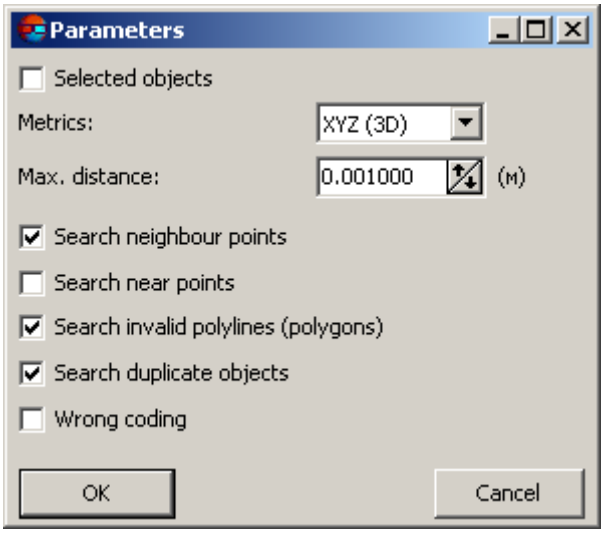
4.4.5.10. *Closing along polyline*

You can create a new polygon, which will have the part of the border topologically coinciding with existing polyline or polygon, using the **Close along polyline** operation.

- place the marker near the vertex of the existing object which will be the last point of the common fragment and press **Insert**.
- continue vectorization by adding new vertices
- place the marker near the vertex of the existing object which will be the first point of the common fragment and press **Insert**.
- Press the **Shift-M** hot keys or select **Edit | Topology | Auto-close along polyline** menu command so that the boundary of the polygon being created will be drawn along existing one within the user-defined extents.
- press **Enter** to complete the operation.

4.4.5.11. *Topology control*

Spatial topological relations between objects imply the coincidence of their common borders during the continuing polyline along another one, as well as the coincidence of vertices during the snapping. In some cases, as a result of operator error the corresponding vertices do not match that is, one vertex can be digitized as two which are very close to each other. The parameters of search process of such duplicate vertices (topology verification) are **Metrics** and **Max. distance** in meters. Pair of points is wrong in that case, if they are from each other closer than **Max. distance** specified parameter. During the topology verification process 2D as well as 3D distances (**Metrics**) can be measured. **Selected objects** option is used to verify topology of selected objects. Above-mentioned parameters of topology verification process are specified in the **Parameters** window which is displayed on **Vectors | Topology | Verify topology** menu command.

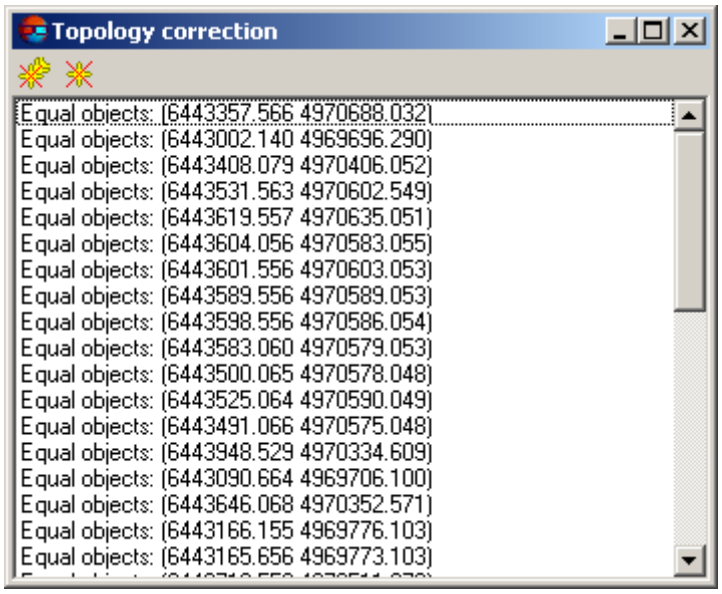


Parameters of topology verification

Set the **Search neighbour points** checkbox to search for coinciding neighbor vertices in a line during the topology verification process.
Set the **Search near points** checkbox to search for coinciding vertices on different lines during the topology verification process.
Set the **Search invalid polylines (polygons)** to search for line objects, which have less than two vertices – for non-closed objects and less than three – for closed objects during the topology verification process.
Set the **Search duplicate objects** to search for line and point objects, which have fully coinciding vertices, during the topology verification process.
Wrong coding option (only applied to vector layers with classifier) allows revealing the point objects matched to linear code and vice versa during the topology verification process. Also revealed are non-closed objects matched to linear code with mandatory closure (code type C).

Upon setting parameters and pressing **OK** button, the topology verification process is started. If errors are detected, a list of coordinates of erroneous points is displayed.

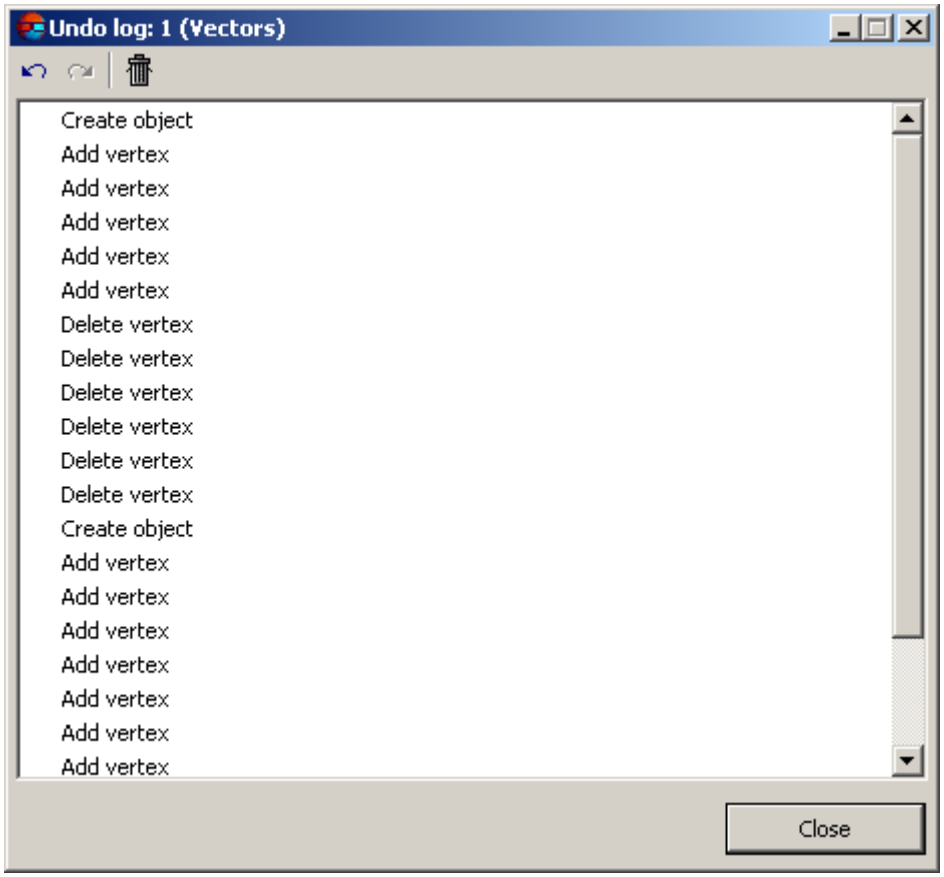
In case any of the above listed errors have been found, the **Topology correction** window opens for review and correction. Each string of coordinates list of erroneous points includes name of error and its plane coordinates.



Selecting point in the list (double click) leads to movement of marker in the appropriate position of 2D window. Erroneous points may be corrected either one by one using snapping or all at once automatically with the **Correct topology** button, or all selected at once with the **Correct topology of selected points** button. As a result of automatic topology correction the erroneous point are shifted to the same point with average values of coordinates.

4.4.6. Undoing the editing operations

To undo the last editing operation, press the button (**Undo last action**) on the toolbar or the **Ctrl-Z** keyboard shortcut. To redo the last undone operation press the button (**Redo last undone action**) or the **Ctrl-Shift-Z** keyboard shortcut. The list of recent editing operations is opened by pressing the button (**Undo log**).



Action Log

To undo the last editing operation, press the button (**Undo last action**) on the toolbar of the window. To redo the last undone operation press the button (**Redo last undone action**). To cancel the group of actions, double-click the left mouse button on the line with the action, to which you want to roll back the changes. Fields with an undone action are marked in italics (see the screenshot). To redo the undone changes, double-click the left mouse button on the field which changes you want to revert. Pushing the **Clear undo log** button leads to the removal list of operations and as result the undo and redo operations become impossible.

There is separate list of actions to cancel for each layer.

Note: While working with a large number of 3D vectors using the undo mode, the editing operations may be performed slowly. To disable the undo mode use the Service | Settings menu command, the Undo section (see also chapter Undo settings).

5. Creation of digital terrain model (DTM)

5.1. *Types of DEM representation*

Main type of digital elevation model is irregular network of triangles – **TIN** (Triangulated Irregular Network). **TIN** may be transformed into regular grid resulting in creation of **digital elevation model** (**DEM**) or represented as a set of **contour lines** (terrain contours, submarine contours or other isolines).
See also the chapter:

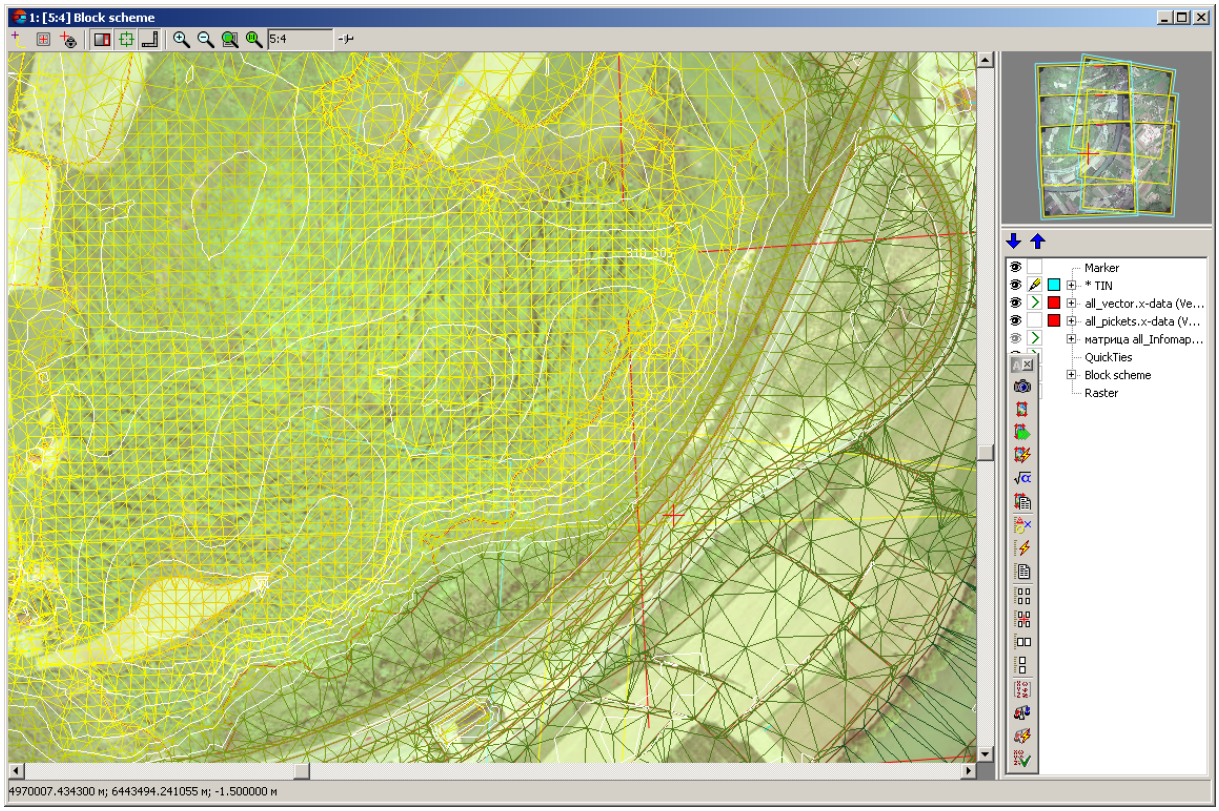
- TIN

- Contour lines
- Digital Elevation Model (DEM).

6. TIN

TIN (irregular network of triangles) in **PHOTOMOD** system is built using base layers, containing vector objects – points, polylines and polygons, and is re-built “on-the-fly” during editing of base vector layers. In addition, during TIN building you can enable creating of quasi-contours – isolines with specified step, passing through TIN triangles edges. Quasi-contours are integral part of TIN and are also rebuild dynamically along with TIN during editing of vector objects of base layers.
I.e. TIN could not exist without “vectors” and it could not be edited (triangles edges and vertices directly), and saved TIN contains just relative paths to base layers, as well as quasi-contours step. When saving/opening TIN, all base vector layers are saved /loaded.

See the chapter Preparing of base layers for TIN creation and Building TIN.



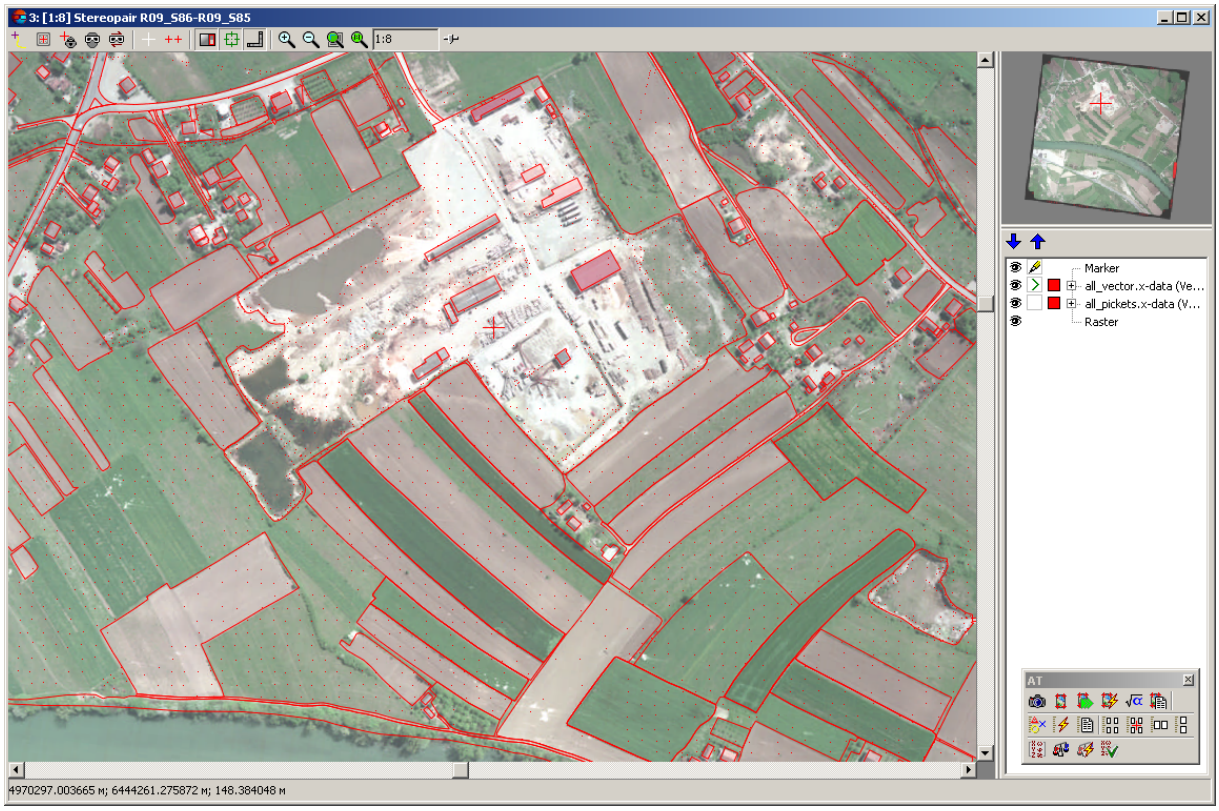
TIN with quasi-contours

6.1. *Initial data for TIN building*

Initial data for TIN building are spatial vector objects (points, polylines and polygons), created using stereovectorization. See the chapter Vector objects. **Pickets** may be also used as base data for TIN, they are point vector objects, without semantic meaning, created on terrain relief using regular grid or in arbitrary order, and covered area selected of TIN creation. See the chapter Pickets creation.

TIN is created considering *all objects of all selected vector layers* (base layers) and is re-build dynamically during any base layers changing – adding/deleting/editing of "vectors" on the layers.

In order to create good TIN it is recommended to use *breaklines* – vector polylines created along continuous relief features (such as ridges or canyons), as well as manmade objects such as roads.



Initial data for TIN building – vector objects

6.2. *Workflow*

There is the following work procedure of creating an irregular network of triangles (**TIN**):
1. Forming of base vector layers. See the chapter Preparing of base layers for TIN creation.
2. Defining of TIN boundaries. See the chapter Building of TIN boundaries.
3. Building TIN. See the chapter Building TIN.
4. TIN viewing and editing. See the chapter TIN displaying, TIN control against triangulation points and TIN editing.

6.3. *Preparing of base layers for TIN creation*

Preparing base layers for TIN creation consists in selection of a set of layers with vector objects (points, polylines and polygons), obtained during the stereo vectorization. See chapter Vector objects creation.

Also the following layers may be used for TIN creation:

- Points - point objects created on relief using regular grid or in arbitrary order. See chapter Pickets creation.
- Breaklines - vector polylines / polygons drawn along the characteristic extended forms of relief (such as shoulders, ridges and thalwegs), as well as along the roads. See also chapter Vector objects creation.
- Triangulation Points - tie/ground control/check points, which are obtained during block measurement stage. See chapter Using triangulation points.

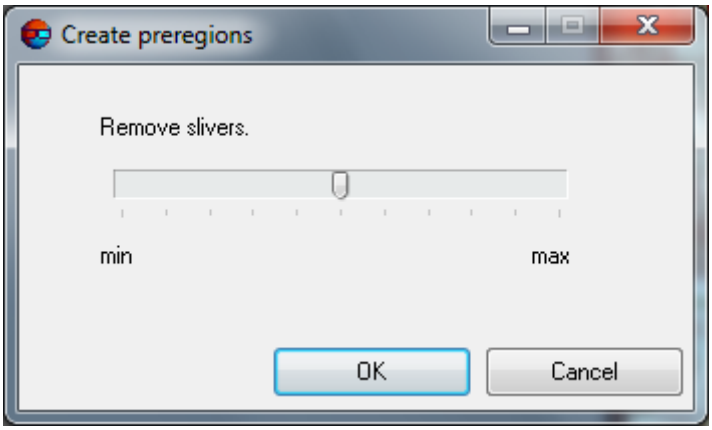
After preparing the base layers, determining borders of TIN, one can proceed to create the TIN (see chapter Building TIN). The TIN will be reconstructed dynamically during any editing of vector objects in layers based on which it has been built.

6.3.1. Pre-regions

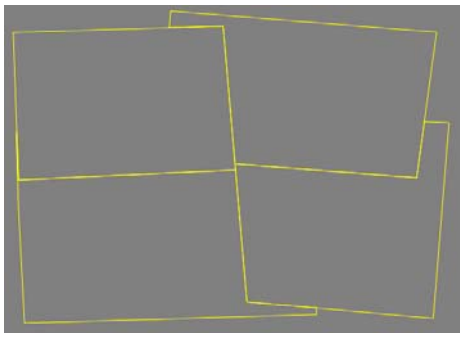
Pre-regions form a vector polygons layer drawn automatically on images block in **PHOTOMOD Core**. Pre-regions allow organizing stereo vectorization of images block in a network. Thus, each operator will work (i.e. create vector objects) in certain area – within pre-region. To do this, open pre-regions (saved as a vector layer) on operator's workstation and define vectorization area.

TINs created for stereopairs can be merged then using pre-regions borders into global TIN for the whole block of images. See the chapter Building TIN.

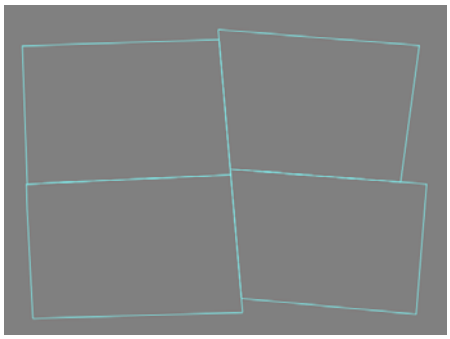
Select menu command **Vectors | Build pre-regions** to create pre-regions. The window used for setting up the only parameter (**Remove slivers**) for pre-regions building is shown.



For examples of pre-regions created at different parameter's values, see figures below.



Pre-regions created when a slider was set to **min** position (without slivers removing)



Pre-regions created when a slider was set to **max** position (with slivers removing)

Pre-regions form a usual vector layer of polygons, which you can edit, save, close and open.

For pre-regions editing use tools of vector objects editing. See the chapter Editing of vector objects.

To save pre-regions make active the *Pre-regions* layer in **Manager** and select menu command **Vectors | Save** or **Vectors | Save as** and save the pre-regions as a usual vector layer in profile resources.
Use menu command **Vectors | Load** to load pre-regions available.
To close pre-regions make active the *Pre-regions* layer in **Manager** and select menu command **Vectors | Close** or right click the *Pre-regions* layer and select **Close**.

6.3.2. Pickets creation

6.3.2.1. *Modes and methods for creating pickets*

Pickets are 3D point vector objects, created on the terrain surface. Pickets, along with other vector layers, are used as base layers for TIN building. Pickets can be created using the correlator. The PHOTOMOD system provides for three modes of pickets creation:

- **Manually in stereo (without or with correlator)**. To add a point, position marker anywhere on the model surface and press **Insert** key (see also chapter Point creation). To add a point with correlator, press spacebar, check correlation result (in the status bar), and add the picket (Insert). Correlator settings are tuned in the **Settings** window (**Service | Settings | Correlator**). See chapter Correlator settings.

- **In semi-automatic pathway mode (without or with correlator)**, which involves preliminary construction of regular grid and passing all the nodes of regular grid one by one for manual pickets adding using the correlator or without, or bypassing nodes (see chapter Creation of pickets in pathway mode). Correlator parameters for the pathway mode are also specified in the **Settings** window. See chapter Correlator settings.
- **In automatic mode using the correlator.** This method also involves creating a regular grid of nodes, in neighborhood of which the correlator tries to calculate spatial coordinates and create pickets. Automatic mode provides wide range of opportunities: setting from one to three nodes processing passes with different correlator parameters, precision control, screening and filtering of pickets, saving quality assessment. Correlator settings for automatic pickets computing are set in the **Compute points automatically** window. See chapter Automatic computation of points.

6.3.2.2. *Grid creation*

Creating pickets automatically with correlator or in semi-automatic pathway mode requires a regular grid with a given step (nodes frequency); spatial coordinates are computed and pickets are created in the vicinity of these nodes. See also chapters Automatic computation of points and Creation of pickets in pathway mode.

The grid can be created on block of images, on any part of the block or on selected stereo pair. Grid border can have any form - rectangle or arbitrary polygon.

Use the **Grid | Create** menu command to create grid (duplicated by the button in the **PHOTOMOD Core** toolbar). This makes a new layer *Grid* to appear in the **Layer manager**. Next, it is necessary to define the *grid area*:

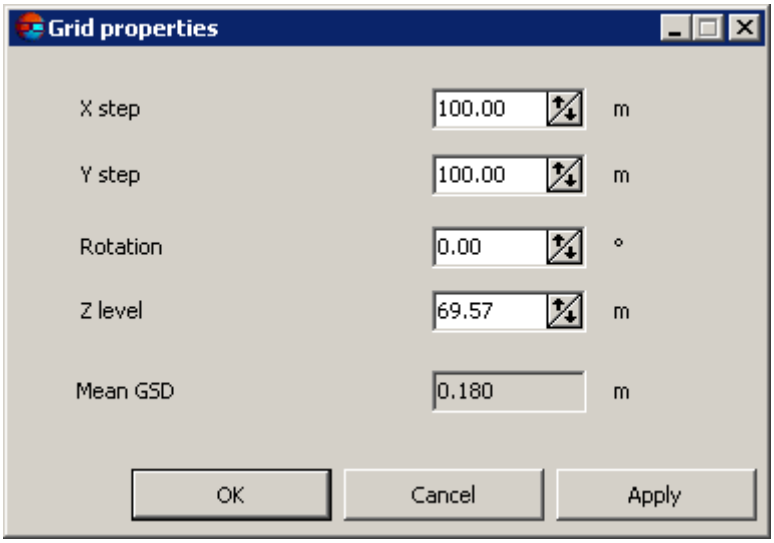
- *To create grid with rectangular frame* draw a rectangular area on the image block or active stereopair with the mouse, holding **Shift** key and left mouse button.

- *You can also create a grid of arbitrary shape*. To do that, use group selection mode by polygon (push the button) and mark the polygon (grid area) apexes by mouse clicks along with **Shift** key pressed down. Complete the polygon creation by double click. Pre-regions may be used for defining the grid area in this way (see chapter Pre-regions).
- *Polygons from a vector layer may be imported as grid borders*. In order to do this, activate vector layer with polygons, select some of the polygons if needed and call menu command **Grid | Create borders from vector layer**. The grid border will be imported from the selected or all polygons of the layer.

- *The grid also can be created for all the images block without explicit borders definition.* In order to achieve this, call menu command **Grid | Properties** without calling or immediately after calling the **Grid | Create** command. Define grid properties (see below) and press **OK**. The grid is created automatically covering all the image block.

If grid borders are specified explicitly, the grid is created immediately after this is done. The parameters of showing grid are could be modified in the Layer manager window. To do this, click the colored square to the left of the layer. **Layer display settngs** window opens, which allows to set the nodes and grid boundaries color, and also the type and size of the symbol to display the grid nodes. Additional settings – showing grid as a semitransparent fill and nodes size – can be changed in the **Settings** window on the **Grid** tab (**Service | Settings**). See chapter Settings of grid displaying.

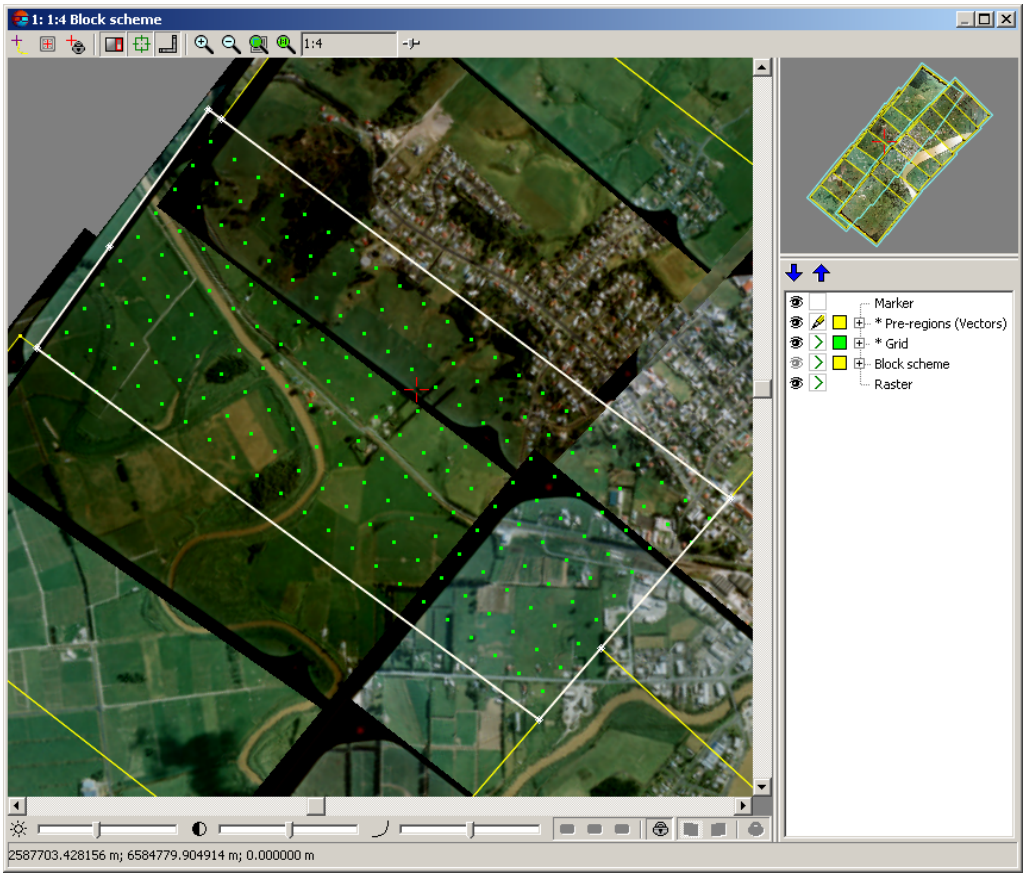
To change the grid properties for the current grid layer, use the **Grid | Properties** menu command. The **Grid properties** window is opened.



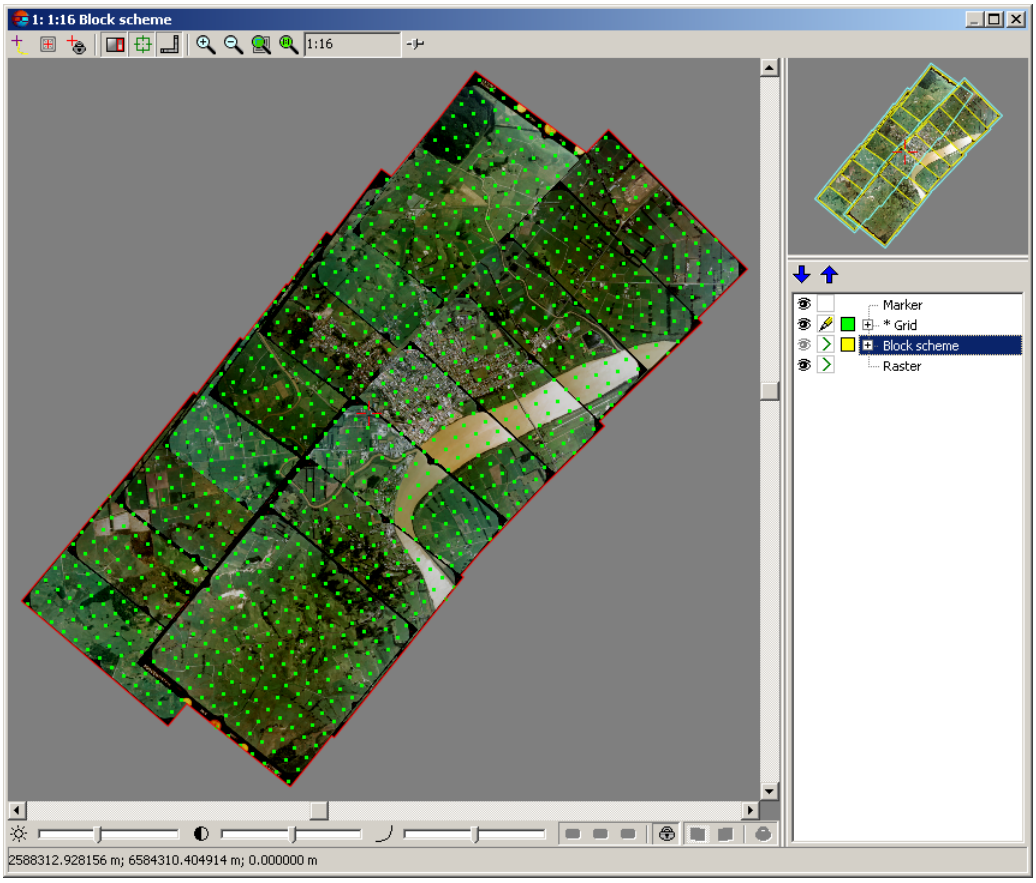
Parameters of regular grid of nodes

Select step of nodes creation by X and Y axes in meters, change grid's rotation angle in degrees if necessary (may be useful at big enough tilt of images relatively to geodetic coordinate system). And specify level of grid in meters by Z (its height over terrain relief). Push **Apply** to view the grid.

© 2011



Grid created from pre-regions, with rotation applied.



Creation of regular grid for the whole images block

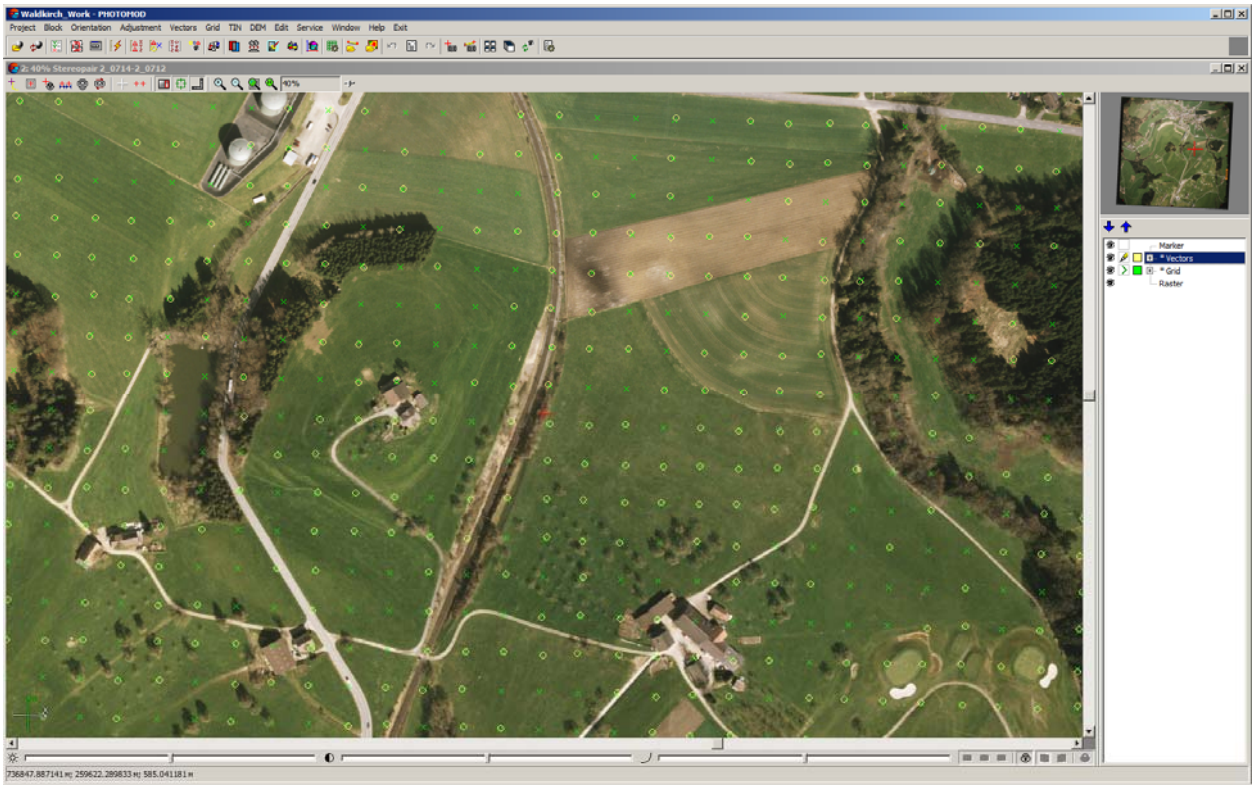
The grid created may be saved in current profile resources, using menu command **Grid | Save** (**Save as**). The grid is saved to file with *x-grid* extension. To open an available grid select menu command **Grid | Load**.

6.3.2.3. *Automatic computation of points*

The system has a capability of *automatic* 3D points calculation in the *regular grid nodes* by using the *correlator* in the images overlap areas

> **Note:** this function is available for oriented stereopairs only. Thus, you have to adjust the block (at least in the free model) before starting points calculation. See the Block adjustment User manual.

The process goes through all grid nodes inside every *oriented stereopair* and tries to calculate Z-coordinate in the point of the best correlation in the node surrounding area. In case of success (all calculated process parameters meet the user-defined acceptable values) the point is added to the output vector layer. Otherwise it is skipped.



Automatic points calculation for selected stereopair

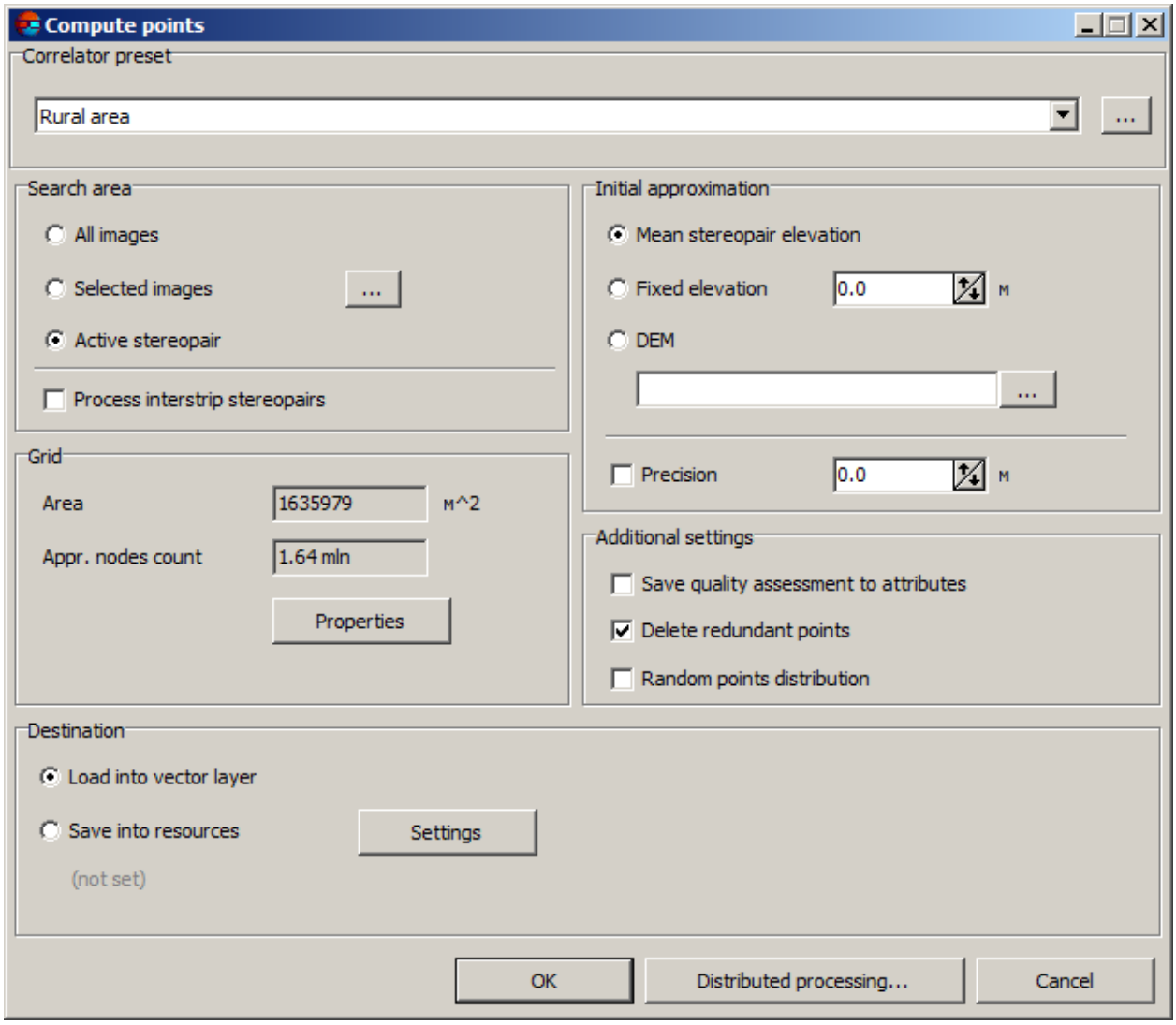
After points automatic computing they can be edited (see the chapter Points editing) and be used as a base layer for TIN creation (see the chapter TIN)

6.3.2.3.1. Computation of points

Before process starting:

1. Define *search area* used for automatic points computation
2. Create regular grid for the *search area* (See the chapter Grid creation)
3. Select images included to the search area in **2D window** or open the stereopair you need (**2D window (stereopair)**)

In order to start the process select **Vectors | Compute points automatically** menu item. If the grid layer was created before **Compute points** window is opened immediately. Otherwise you will need to setup the grid parameters in the **Grid properties** window (see the chapter Grid creation). In this case the grid covers the images block totally.



Compute points window

Compute points window consists of following parameter panels:

Parameters	User actions
Correlator presets used to setup the correlator in accordance with the type of territory and type of sensor	Following pre-defined presets are available: • **mountainous terrain** • **mountainous terrain (satellite)** • **urban area** • **rural area** Use button ... to view or edit the preset parameters or create or own preset. See the chapter Preset editing.
Search area used to define process area	Following options are available: • **All images** - to use all block images; • **Selected images** - to use images selected in **2D window (block)**; • **Active stereopair –** to start process for active stereopair – stereopair opened in **2D window (stereopair)**.

	Use button ... to select another group of block images. By default the process works for in-strip stereopairs. And in very most cases it is enough to get good results. Use option **Process interstrip stereopairs** if you want to include them to process.
Grid used to view and edit the grid parameters.	Following grid parameters are displayed: • **Area** (sq. meters.); • **Approximate nodes number**. Use **Properties** button to change grid parameters (see the chapter Grid creation).
Initial approximation used to define the *initial approximation* for the correlator. This parameter sets up the approximate value of Z-coordinate for every grid node used as a first approach for searching the point on the left and right stereopair images	Following options are available: • **Mean stereopair elevation.** Calculated from the exterior orientation parameters; • **Fixed elevation**. User-defined Z-value; • **DEM**. DEM selected by user. For possible NULL-cells the mean stereopair elevation is used. Use button ... in order to select the DEM in the active profile. You can also set the deviation in meters from the initial approximation model by using the **Precision** parameter.
Additional used to setup additional process parameters.	Following options can be used if needed: • **Save quality assessment to attributes** (see Note 1 in this chapter). • **Delete redundant points** to thin out the points obtained from different stereopairs on the same grid nodes and, thus, located very close to each other. Process takes into account the minimum distance between the points defined in the current preset parameters (see the chapter Preset editing). If the option **Save quality assessment to attributes** is on the system will keep the point with the highest correlation value (**"corr"** attribute) and deletes the rest of points. • **Random points distribution** causes random points location in the area of ½ grid cell size.
Destination used to load or save calculated points	Available options are as follows: • **Load into vector layer** to display the output points. If there is no vector layer loaded it is created. • **Save into resources** to save output points to vector resources without displaying. Use **Settings** button to select output resources. See Note 2 of this chapter for details.
Distributed processing, OK, Cancel buttons	Push **Distributed processing** button to start points computation in distributed processing mode. It is very useful for big projects. See the chapter Computing points in distributed processing mode. Use **OK** button to start processing and **Cancel** button to stop it.

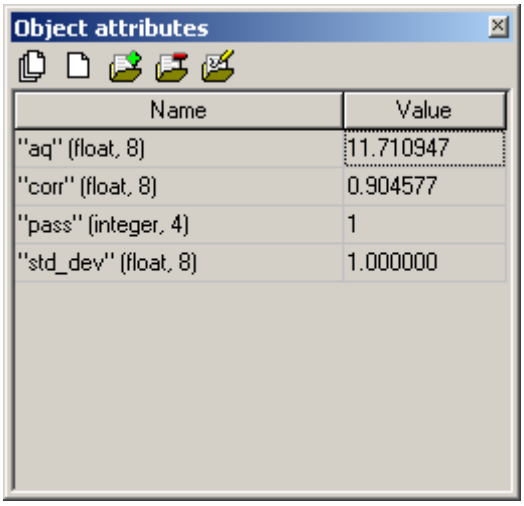
Note 1

Save quality assessment to attributes option allows saving the following attributes (regardless of classifier been assigned to vector layer or not, see chapter Vector objects:

- **"corr"** (double) – correlation coefficient, a floating point number in range [0;1] (with normalized correlation threshold been the actual lower boundary);

- **"aq"** (double) – autocorrelation sigma;
- **"std_dev"** (double) – nominal std. deviation of point geometry, equal to relative pixel size of the pyramid level chosen for the correlator pass which resulted in the point (i.e. it is 1 for the zero pyramid level, 2 - for the first one, 4 for the second, etc.). It actually reflects accuracy of the measurement;
- **"pass"** (int) – correlator pass number which resulted in the point measurement, may be used for further analysis of preferable strategy for the given relief type (equals 1,2 or 3).

In order to view attribute values you should select the point and open **Object attributes** window (**Windows | Object attributes**).



Quality assessment for a computed point

Note 2

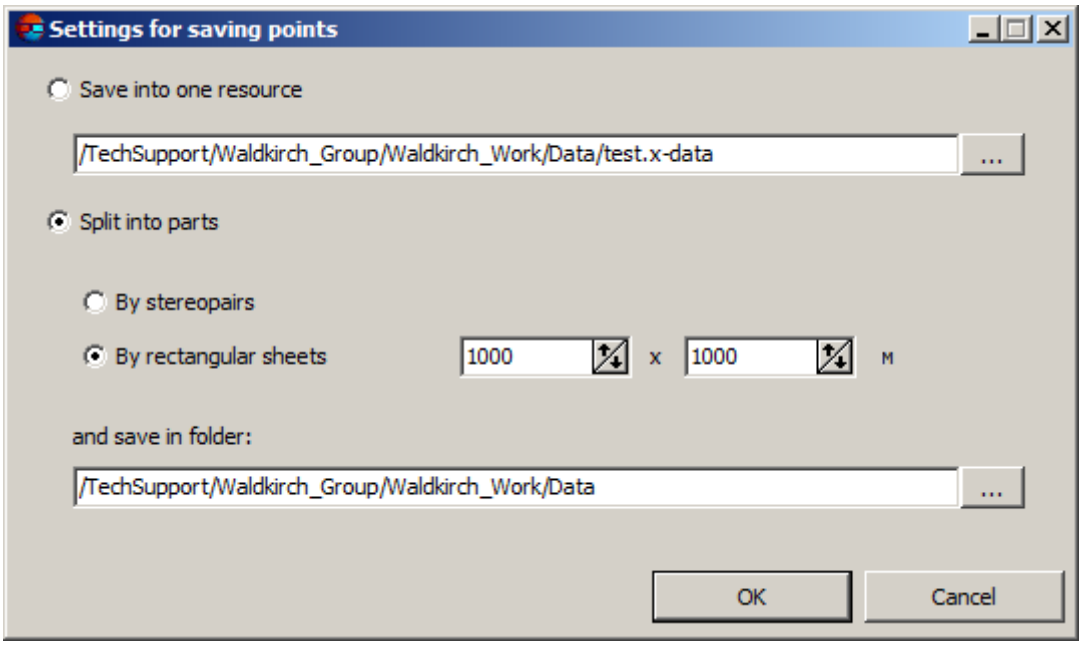
Settings for saving points window allows to save points into one output resource or distribute them into several resources of the active profile.

In order to save points into one vector resource of the active profile you should:

1. Select **Save into one resource** option and push ... button.
2. Select the folder in the opened **Open** window and enter the resource name in the **Resource name** text field. Push **OK** button.
3. As a result the output path and resource name are displayed in the appropriate text field. Push **OK** button to get back to the **Compute points** window.

In order to save output points into several vector resources you should:

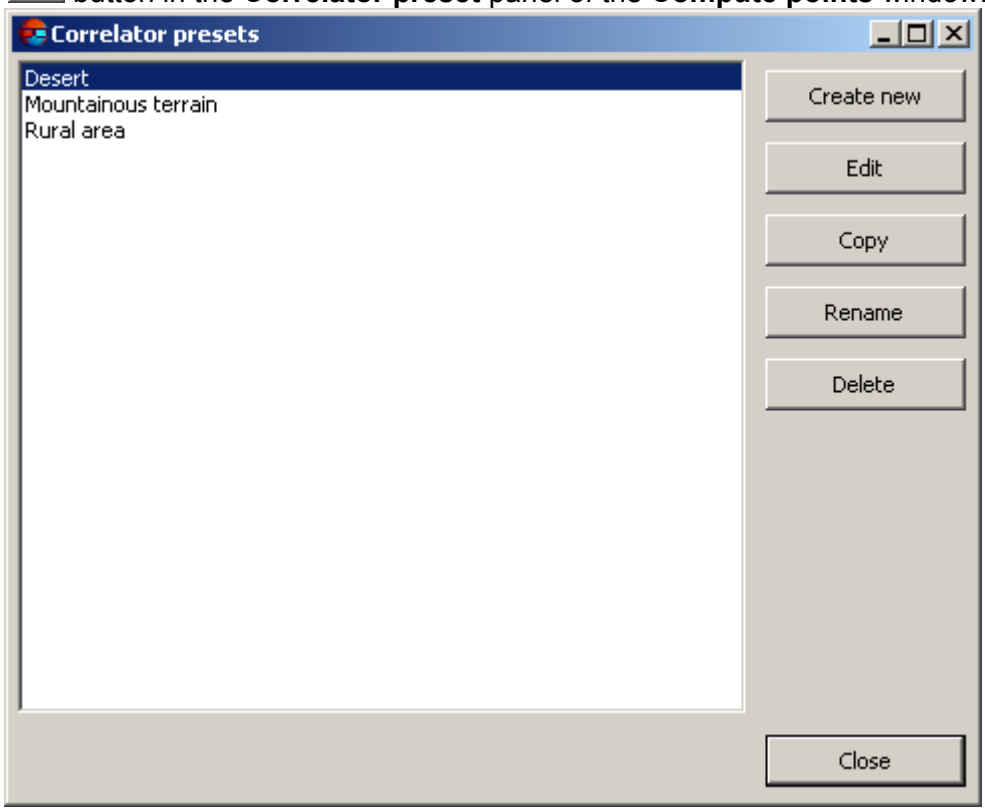
1. Select **Split into parts** option.
2. Choose between **By stereopairs** and **By rectangular sheets** options depending on how do you want to divide the points. In second case set the size of the rectangular area in meters.
3. Click ... button rightward to **and save in folder** text filed and select output folder to be used for saving resources.
4. As a result the output path is displayed in the appropriate text field. Push **OK** button to get back to the **Compute points** window.



Settings for saving points window

6.3.2.3.2. Correlator presets

Correlator presets window is used to form the *correlator presets* list. To open the window push the ... button in the **Correlator preset** panel of the **Compute points** window.



Correlator presets window

The list of all created *correlator presets* is displayed in the left side of current window. The list contains the standard *correlator presets* by default with parameters tuned for the following terrain types:

- **Mountainous terrain**
- **Desert**

- **Rural area**

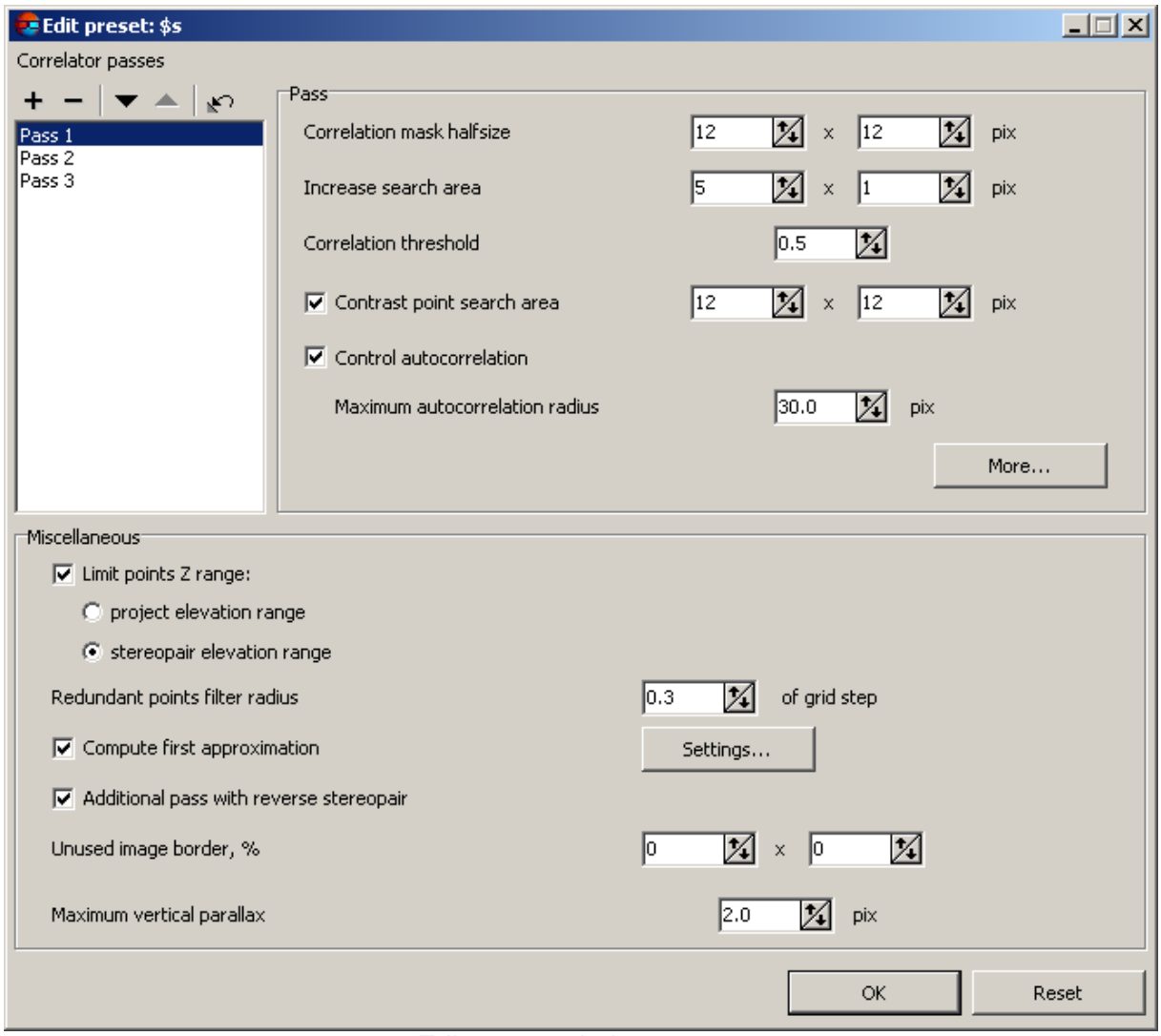
The right panel of the window contains the following buttons for managing the *correlator presets*:

Button	Description
Create new	Creating new correlator presets. The **Create preset** window opens by pushing the appropriate button. Enter the new preset name and push the **OK** button. The **Edit preset** window opens. Edit the parameters and push the **OK** button. New preset is saved and displayed in the list of **Correlator presets** window.
Edit	Editing correlator preset selected in the list. Select the preset in the list and push the **Edit** button. The **Edit preset** window opens. Edit the parameters of the selected preset and push the **OK** button to save the changes made.
Copy	Copying correlator preset selected in the list. Select the preset in the list and push the **Copy** button. The **Create preset** window opens. Enter the new preset name and push the **OK** button. The copy of selected preset appears in the list of **Correlator presets** window.
Rename	Renaming correlator preset selected in the list. Select the preset in the list and push the **Rename** button. The **Create preset** window opens. Enter the new preset name for selected preset and push the **OK** button. The new name of selected preset is displayed in the list of **Correlator presets** window.
Delete	Deleting correlator preset selected in the list. Select the preset in the list and push the **Delete** button. The deletion request of selected preset is appear. Push the **OK** button. The selected preset will be deleted from the list of **Correlator presets** window.

6.3.2.3.3. Settings of correlator preset

The **Edit preset** window is used to viewing and setting the correlator preset, which is available for opening in following cases:

- Editing of current preset. Select the preset in the list of **Correlator presets** window and push the **Edit** button.
- Creating of new preset. Push the **Create new** button. Enter the new name of preset and push the **OK** button.



Edit preset window

Editing of *correlator preset* in the **Edit preset** window consists in defining scenario of automatic points computation. Depending on the relief the scenario can include one or several grid node *passes*, for each of which the parameters set is configured.

The **Correlator passes** list displays the created *passes* set with specified correlator parameters. The passes in the list are located in order of their usage during the automatic computation of points. The following buttons are provided to form the passes list:

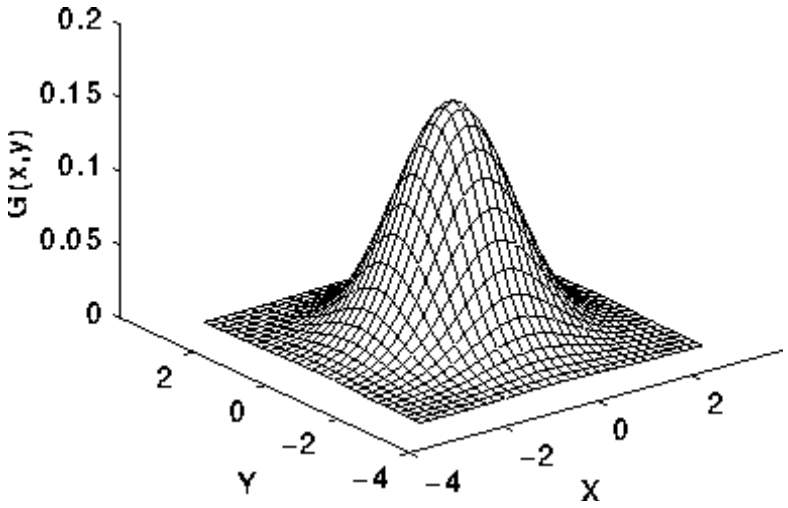
Button	Description
New pass	Add the new pass to the end of list
Delete pass	Delete selected pass in the list
Move further to end	Move selected pass in list to bottom
Move closer to start	Move selected pass in list to top
Reset pass settings to defaults	Reset pass settings to default

The main correlator parameters for selected pass from the list are specified in the **Pass** panel. **Pass** panel contains the following parameters:

© 2011

- **Correlation mask halfsize** – half of correlation mask linear size in pixels along X and Y axes (the mask has a rectangular form). Default value - 12x12 pixels. Increasing the size of the mask increases the recognition accuracy for smoothly varying terrain and reduces the impact of small parts on the surface, but reduces the total outcome of points. It should also be noted that working on material with low-contrast areas, corresponding to oblique forms of relief (for example, fields or lawns on the hills), points may be added corresponding not to the center but the edge of the mask.
- **Increase search area**. Parameter is intended to remedy the situation, when estimated search area of the corresponding point on the right image derived from the orientation data of the stereo pair is too small (e.g. not enough tie points on high-altitude objects or in the corners of a stereo pair). Typically, in this case we are talking about tens or hundreds of pixels (along X axis). The requirement to increase the search area along Y can occur when the accuracy of the relative orientation stereo pair is low.
- **Correlation threshold** – the minimum allowed value of the correlation coefficient. The value essentially depends on the quality of the material, as well as on other parameters. However, it is generally recommended to set the value not less than 0.7.
- **Contrast point search area**. Determines the size of the area (in pixels) around the initial approximation (a grid node), in which point of maximum contrast is searched for before the correlation is performed. It is recommended to increase the value for images with clear contours of objects, reduce for low-contrast areas, where the probability of correlator error is high.
- **Control autocorrelation** and **Maximum autocorrelation radius**. Option allows to control the autocorrelation of point, i.e. degree of point uniqueness within some neighborhood on the left image. The higher the value of the autocorrelation radius, the less precise is the point and the greater the likelihood of incorrect matching with the right image, even with high correlation coefficient.

  Note:
  The autocorrelation is controlled as follows: the two-dimensional function of correlation of the mask with the regions on the same image with shift relative to the mask up to given number of pixels (the maximum is obviously equal to 1 at point with zero displacement), and then the coefficient σ is calculated in the formula for two-dimensional surface, which is described by a Gaussian function: $f(r) = e^{-\frac{r^2}{2\sigma^2}}$, where r - the shift of the area relative to the mask, so that the $f(r)$ graph describes the distribution of the correlation coefficient above. Thus, σ - radius of the area, beyond which the correlation mask coefficient is not greater than 0.5. The smaller σ, the more sharp peak forms by the autocorrelation function at 0, and the more the mask is precise and the less likely that the area on the right image with the maximum correlation coefficient actually corresponds to the shifted mask.



Filter by the autocorrelation radius rejects the point which having the parameter σ greater than this radius. Thus, in particular, points on vector objects are "rejected".
On the contrast material (such as urban area in the serene atmosphere at digital imaging) values of radius may be set to 8-9 pixels, in case of survey with a "smear" (or in the case of rural areas on the aerial survey) - about 30 pixels.

More button allows to configure additional settings for the selected pass. See the description in chapter More settings of pass.

Miscellaneous panel contains the following options:

- **Limit points Z range**. Parameter specifies the criterion of rejection by Z-range, which is applied to each calculated point. Set the radio-button to one of the following positions:
  – **Project elevation range**, defined by user in the project properties;
  – **Stereopair elevation range**, calculated during the adjustment of block images.
- **Redundant points filter radius**. The option is used to filter adjacent pickets created by passing through the same grid nodes on different stereo pairs. The thinning of pickets is carried out after pickets computation is complete, based on the specified radius in meters.
- **Compute first approximation**. Option allows to perform a preliminary calculation of the pickets, which settings are specified using the **Settings** button (it is located on the right). A preliminary calculation is recommended to perform for the scanner imagery. See the description in chapter Preliminary calculation settings of the first approximation.
- **Additional pass with reverse stereopair**. Option enabled the additional pass during the computing of the pickets, using which the search is done for the corresponding points on the left image for points on the right image (during the main pass the points on the right image which correspond to the points on the left image are searched).
- **Unused image border, %**. Option allows to not take into account during the computing of the pickets area of image.
- **Maximum vertical parallax**. Option is designed for rejection points to set limits on the vertical parallax (in pixels).

  Note:

  The point generated by correlation has 4 coordinates - XL, YL on the left image and XR, YR, on the right image. To calculate the vertical parallax is performed the coordinate transformation from image to the coordinate system of the project and vice

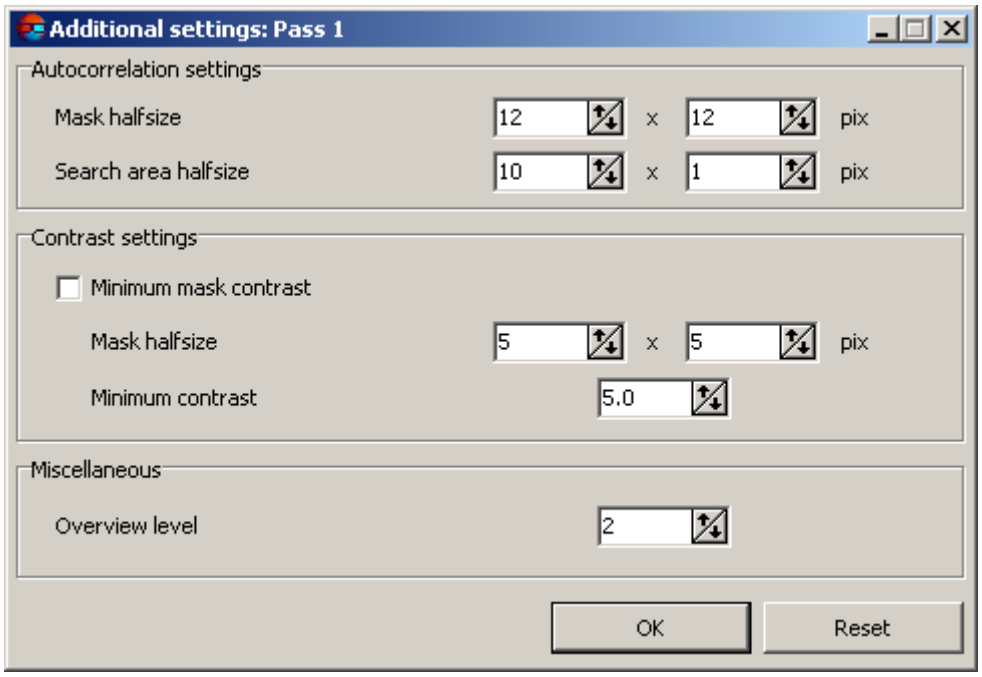
versa, two pairs of new coordinates (XL2, YL2) and (XR2, YR2) are obtained as the result, and then the parallax is calculated as:

YPAR= (YR-YL)-(YR2-YL2),

and the point are rejected with the value | YPAR | greater than the specified value. This option should be selected experimentally, especially in the case of scanner imagery.

6.3.2.3.4. More settings of pass

To configure the additional correlation parameters for the selected pass in the **Edit preset** window the **More** button is used, which opens the **Additional settings: Pass №** window.



Additional settings for selected pass

Autocorrelation settings panel allows to set the additional settings for autocorrelation:

- **Mask halfsize** – half of correlation mask linear size in pixels along X and Y axes. Default value - 12x12 pixels.
- **Search area halfsize** – half of area size, in which autocorrelation surface is created.

Contrast settings panel allows to take into account the contrast for low-contrast images. Check the **Minimum mask contrast** option and set the followings settings:

- **Mask halfsize** – half of area size around of point, in which the contrast calculation is carried out (rms deviation of points brightness of the mean).
- **Minimum contrast** – the value in a range of 0 to 256.

Overview level parameter allows to perform the correlation between the subsampled images. 0 value corresponds the based level of epipolar images (it is corresponds to 1:1 zoom in stereo window), 1 – subsampled twice, 2 – subsampled fourthly, etc. Using the **Overview** parameter useful in the following cases:

- On the grainy film imagery, which interfere with the reliable operation of the correlator.
- On the relief surface of type the field, which has in a base detail level the random irregular items (grass and small shrubs), or conversely the regular figure (a freshly plowed field), on which the comparable textures are appear on the left and right images if the scale is decreased.

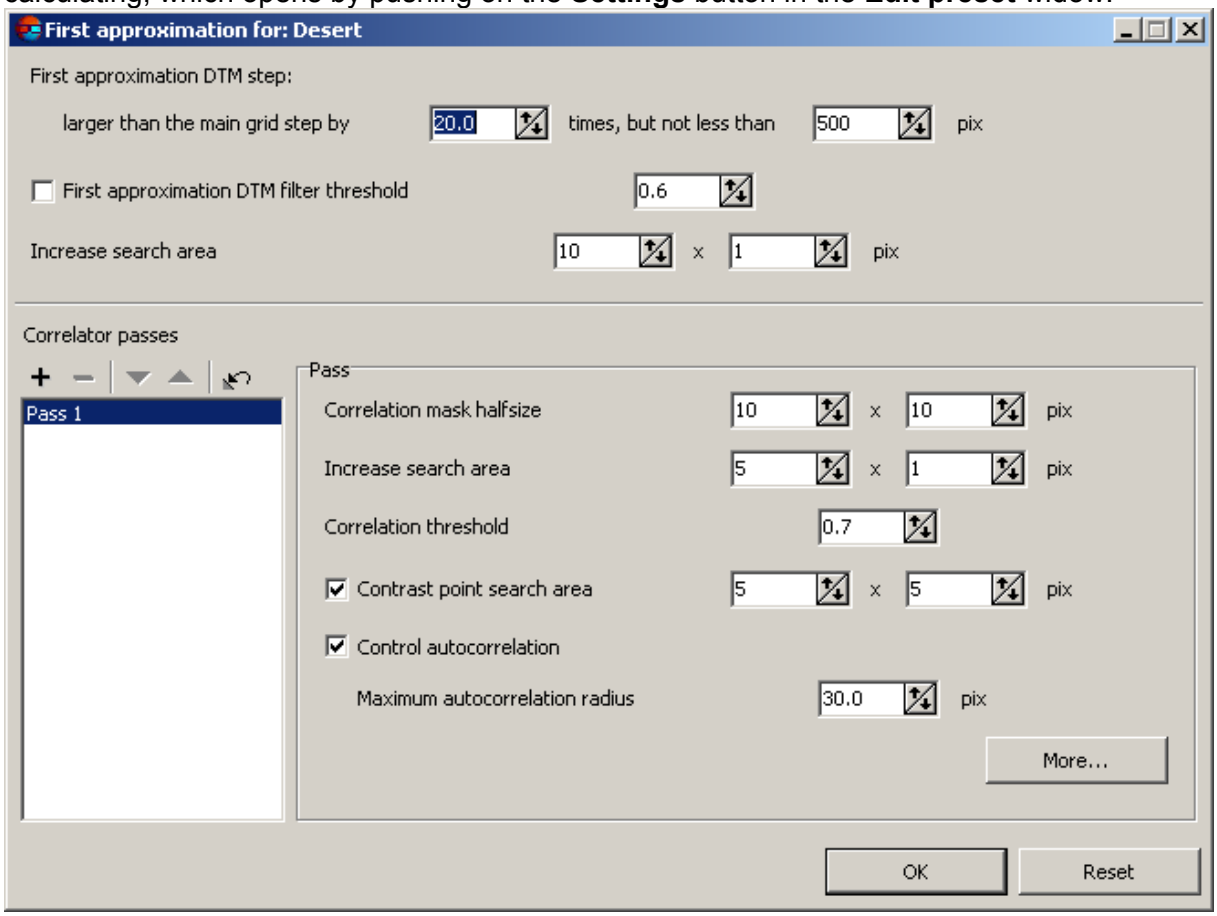
- In cases when it is necessary to exclude small objects on the relief (bushes / separate trees / houses). In this case, subsampling jointly with increasing the mask size leads to the fact that, even if the center of mask falls on the outshot object, the resulting point elevation corresponds to the surrounding terrain.

In any case, the increase percentage of good point obtainment is accompanied by a decrease of accuracy in proportion to the used scale (twofold to first overview level, etc.). Nevertheless, for the relatively flat relief, or not too high demands on result accuracy, this "diminution of accuracy" can be justified.

It should be used the correlator settings, which differ by no greater than one level of the overview (for example, 0 and 1; 2 and 3) as the passes of the same calculation process, otherwise, in areas with small details (such as houses), may appear the «staggered» grids of interleaved points at the height of the object and at the height of the terrain.

6.3.2.3.5. First approximation settings

"DTM first approximation" is represented by a TIN built over the subsampled set of points. Creating such DTM allows to narrow search area for each point of the main process and thus reduce the total search time and likelihood of appearing of erroneous points.
First approximation for window is used to set the process settings of first approximation calculating, which opens by pushing on the **Settings** button in the **Edit preset** widow.



First approximation settings

First approximation for window allows to specify the following parameters:

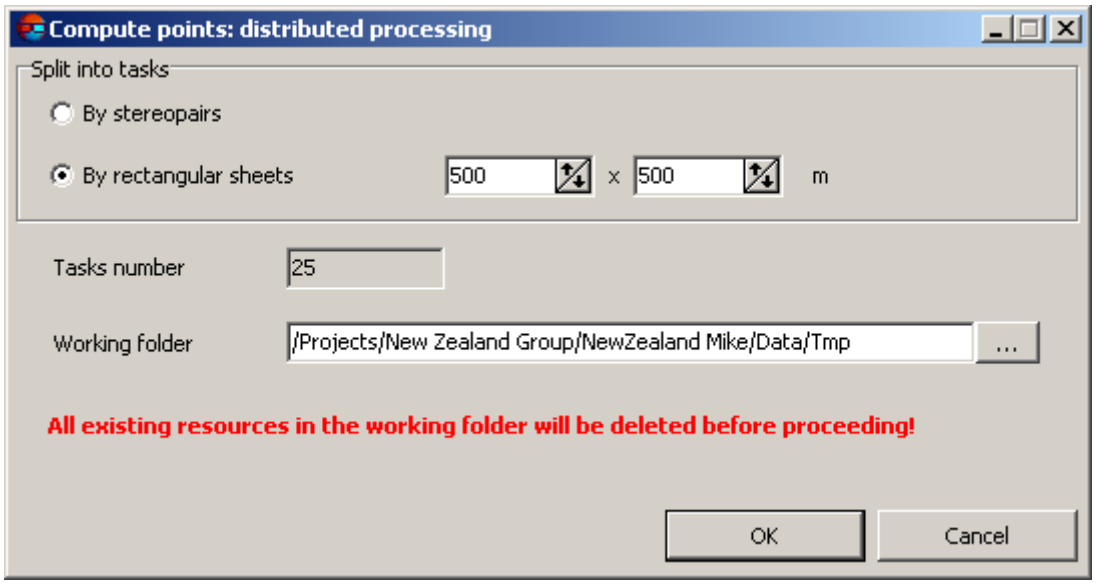
- **First approximation DTM step** – to specify DEM step;
- **First approximation DTM filter threshold** – the TIN filter threshold for the minimal angle of slope of a normal, normalized to the range (0; 1) (see chapter about TIN filtering);

- **Increase search area** – additional increase of search area beyond values specified by the correlator pass parameters.

The list **Correlator passes** and parameters in the **Pass** panel allow to specify the correlator preset for preliminary pickets calculation. See description of main parameters in the chapter Settings of correlator preset. The **More** button serves to specify additional parameters for the pass selected. See description in chapter More settings of pass.

6.3.2.3.6. Computing points in distributed processing mode

For starting points computation in distributed processing mode, press the **Distributed processing** button in the **Compute points** window (**Vectors | Compute points automatically** menu command). The following window opens:



Note:
Each distributed processing task saves its result in a separate resource in the specified working folder of the profile resources. As the corresponding warning message states, the working folder is cleared (including deleting of all subfolders) when the tasks are posted. So using any project folder with existing data leads to loosing all this data, and is strictly not recommended, especially when there are no backup copies of such data.

The computing job may be split in distributed processing tasks in two ways:

- **By stereopairs**. Each stereopair is processed as a separate task. It is supposed that the data would be further edited manually (in stereomode). Points computed for the same grid nodes from different stereopairs are not filtered, because tasks work independently of each other. The computed points are saved in resources named after the stereopairs in the working folder.
- **By rectangular sheets**. The rectangle circumscribed around the grid area in the coordinate system of the project, is split into rectangular sheets of the given size; each sheet is collected from all stereopairs it intersects with. Adjacent points are filtered. The result is saved for each sheet in a resource named Sheet_X_Y.x-data where X and Y are sheet numbers along X and Y axes respectively. This strategy implies further automated processing, e.g for batch creation of DEM (see chapter Batch DTM creation).

6.3.2.4. *Creation of pickets in pathway mode*

Pickets could be collected in semi-automatic way with linking their XY coordinates to regular grid nodes on a stereopair, using **Pathway mode**.

In this mode the operator can set Z coordinate of each picket manually or using correlator. After that the operator adds or skips the picket in current grid node and passes to the next one. During marker positioning in first node the initial Z coordinate of marker is setup regarding to grid Z-level specified in the **Grid properties** window, see the chapter Grid creation. Marker Z level after jumping to subsequent node is defined by settings (see below).

Adding points in pathway mode is performed in *stereo mode* in the *stereopair 2D window*. At that, a global grid for all the project may be used, and automatic switching between stereopairs during passing through grid nodes may be set up.

Preliminary settings are defined in the **Settings** window (menu command **Vectors | Pathway | Settings** which opens **Pathway** mode tab of the Settings window). This window allows setting Z level for marker after jumping from node to node, limit grid traversing by the active stereopair regardless of the grid limits, or on the contrary traverse all the grid with automatic stereopair switching in 2D window. See detailed description in chapter Pathway mode settings.

Before starting pathway mode, grid for the area of interest and target vector layer (new or with existing data) for points could be created beforehand. If the necessary layers have not been prepared before, they are created automatically when pathway mode is activated (see below).

To start creating points in pathway mode:

- *Turn the pathway mode on*. Execute **Vectors | Pathway | Activate** menu command. A grid layer and a vector layer are created automatically in case they have not been created before, the vector layer is made active and marker moves to the starting node (*lower left grid node in the project coordinate system*).
- *Define the traversing direction* of the grid with **Vectors | Pathway** menu command or **Pathway mode** toolbar (see details below): scan grid rows or columns, or "zigzag" over rows or columns. Besides, more control over grid traversing mode is given by possibility of negative values for grid step (by X,Y) in the Grid properties window (see chapter Grid creation).
- *Open a stereopair*, using the main toolbar button (**Open new 2D window for the selected stereopair**). Turn on the stereo mode with the (**Toggle stereo**) button on the 2D window toolbar. make sure the 2D window has focus, and points entering mode ( ) is chosen in **Vectors** toolbar.
- *Enter the points.* In pathway mode, points are added by pressing **Enter** key. Points can be settled on surface with correlator, by pressing **Spacebar** key. If the correlation succeeds, the status bar displays correlation coefficient, otherwise it displays "bad point" warning. Press **Enter** to add the points. Press **Delete** to bypass the grid node without adding point and jump to the next node. Correlator parameters, as well as beep on unsuccessful correlation attempt, are set up in the **Settings** window (**Service | Settings | Correlator**), see chapter Correlator settings. To return to the previous grid node, use the **Backspace** key. Last added point may be deleted with **Ctrl-Z** shortcut – the standard undo function (see chapter Undoing the editing operations), if undo is active in the system settings (see chapter Undo settings).

To turn the pathway mode off, call the **Vectors | Pathway | Activate** command once more (the menu entry is unchecked).

Current grid node is saved when the grid is saved, so pathway grid traversing may be interrupted and afterwards restarted from the same node.
After all grid nodes have been passed, the "**Pathway complete**" message is displayed. The vector layer may be saved with the standard **Vectors | Save** command.
The following commands in Vectors | Pathway menu and Pathway mode toolbar are available for the pathway mode:

- **Activate** ( ) – toggle pathway mode on/off.
- **Add point** (**Enter** key) – add a point in current marker position.
- **Bypass point** (**Delete** key) – bypass node (without adding point) and jump to the next node.
- **Return** (**Backspace** key) – return to the previous node (without deleting point).
- **Marker to current grid node** ( ) – move marker to the current grid node.
- **To nearest grid node** ( ) – move marker to the nearest grid node and make it current. This command works only when marker is within grid border (in project coordinate system), and is designated for setting pathway starting node manually.
- **To first grid node** ( ) – move marker to the first (lower left) node and make it current.
- **Mode: scan rows** ( )
- **Mode: scan columns** ( )
- **Mode: "zigzag" by rows** ( )
- **Mode: "zigzag" by columns** ( )
- **Settings** – open the system Settings window on the **Pathway mode** tab, See chapter Pathway mode settings.

6.3.2.5. *Points editing*

- **Points deleting, adding, moving**

Operations of points editing are applicable to selected point or group of points. Selecting and editing points requires activating vector layer containing points in **Layer Manager**. Double click the point to select it or use the **Selection modes** toolbar to select a group of points. Since points are vector objects you can apply all operations of vector editing to them. See also chapters Point editing and Objects selection.

To add a new point to active vector layer, it is necessary to set point entering mode (**Edit | Vectors create mode | Points**), place the marker into needed position and press **Insert** key.
To add new point using correlator (on the stereopair), use **Spacebar** to assess the correlation (shown in status bar). Press **Insert** if the correlation result is satisfactory. Parameters and sound for correlation are set up in the settings window (**Service | Settings | Correlator**). See chapter Correlator settings.
Also there is a command **Vectors | Operations | Delete points around polylines**, which allows (for several layers as a one) deleting all points located within the given range from polylines. See chapter Deleting pickets around line objects.

- **Filtering points**

There are several filters available for point objects:
Filter by Z-range – filter out *points and polyline/polygon vertices*, Z coordinate of which falls outside the given absolute range.

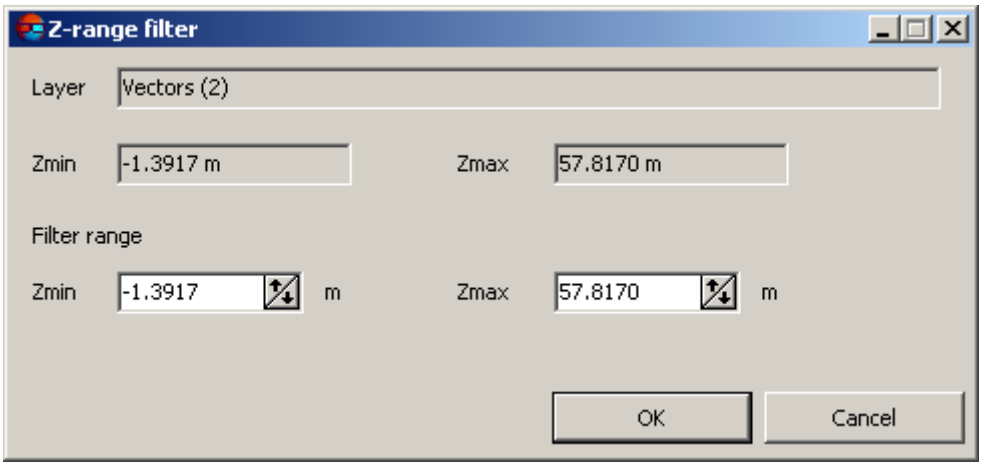
Median Z filter – filter out *points and polyline/polygon vertices*, Z coordinate of which falls outside the given range from median Z value for a certain area.
Filter adjacent point objects – filter out *points* situated closer than the given range from one another.
Filter of buildings and vegetation – filter *points* set on elevated objects (buildings, trees) or in pits, to obtain layer of points describing the relief.

- **Verifying points topology**

Vectors | Topology | Verify topology command may also be used for finding points coinciding with line object vertices. See chapter Topology control.

6.3.2.6. *Filter by Z-range*

For filtering objects of the active vector layer, execute the **Vectors | Filter | Filter by Z-range** command. The **Z-range filter** window opens:

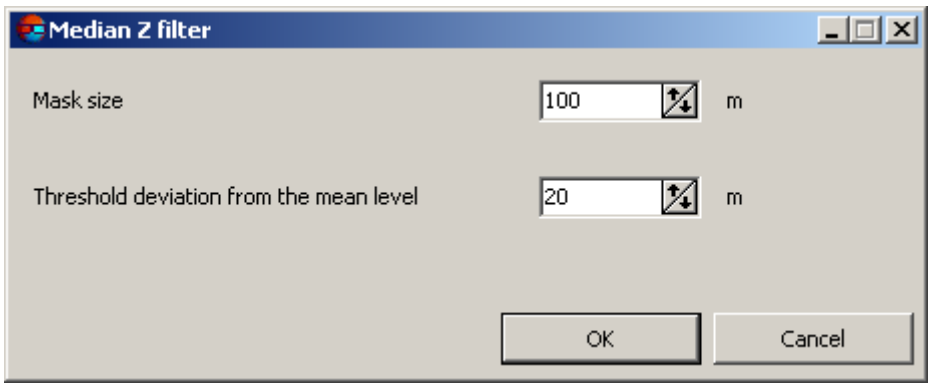


The window displays the name of the active vector layer and range of Z coordinates for all objects in the layer. This range also is used for default values in the **Filter range section**. Adjust these values and press **OK**. All points and polyline/polygon vertices having Z coordinates outside the specified range, are deleted.
Z-range filtering is also available in automatic points computing (see chapter Computation of points.
See also chapters
Median Z filter
Filter adjacent point objects
Filter of buildings and vegetation.

6.3.2.7. *Median Z filter*

This filter consists in sequential scanning of the vector layer area by a window (mask) of the given size (with step equal to ½ of the mask size) and deleting points and vertices, Z coordinate of which falls outside the given threshold from the median Z value of all objects inside the mask. This filter suits for cleaning points layer from sharp surges on the smooth relief.

To apply the filter, execute the **Vectors | Filter | Median Z filter** command. The corresponding window is opened:

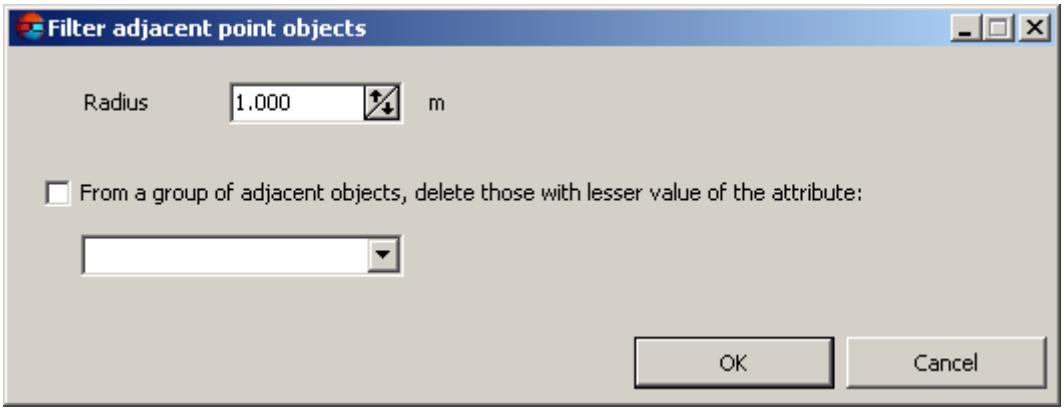


Set the **Mask size** – size of the scanning mask in project units (usually meters), **Threshold deviation from the mean level** – allowed deviation of altitudes from the mean Z coordinates of objects inside the mask. Press **OK**.

See also chapters
Filter by Z-range
Filter adjacent point objects
Filter of buildings and vegetation.

6.3.2.8. *Filter adjacent point objects*

Filter of adjacent point objects erases point objects which are located closer than the given distance from other objects. Make the vector layer with points active and execute menu command **Vectors | Filter | Filter adjacent point objects**. The window opens:



Set **Radius** – the distance in project units (meters), specifying minimum allowed distance to neighbor points.
If the vector layer has been created with automatic points computing, *with saving quality assessment data into attributes*, it is reasonable enabling option **From a group of adjacent objects, delete those with lesser value of the attribute** and choosing the attribute which would be considered during filtering. See chapter Computation of points (extended mode).
See also chapters
Filter by Z-range
Median Z filter
Filter of buildings and vegetation.

6.3.2.9. *Filter of buildings and vegetation*

This buildings and vegetation filter is used for removing, correcting or just detecting points measured on buildings, trees, cars, in pits, etc. after automatic points calculation (see chapter Automatic computation of points). Random surges are also filtered out. Only points describing the relief are left.

We further call all points not measured on relief as "surges".

It is recommended (as default) to apply three filter stages in sequence (three passes):

- **Basic filter** – for removing points standing out significantly from adjacent points. These may be points on freestanding trees as well as in forest areas. This stage also can detect majority of points on buildings as well as correlator blunders (i.e. sharp surges);
- **Additional filter** – filter points not consistent with adjacent points and missed on the previous stage;
- **Detailed filter** – remove points on low-height objects (small buildings, cars, etc).

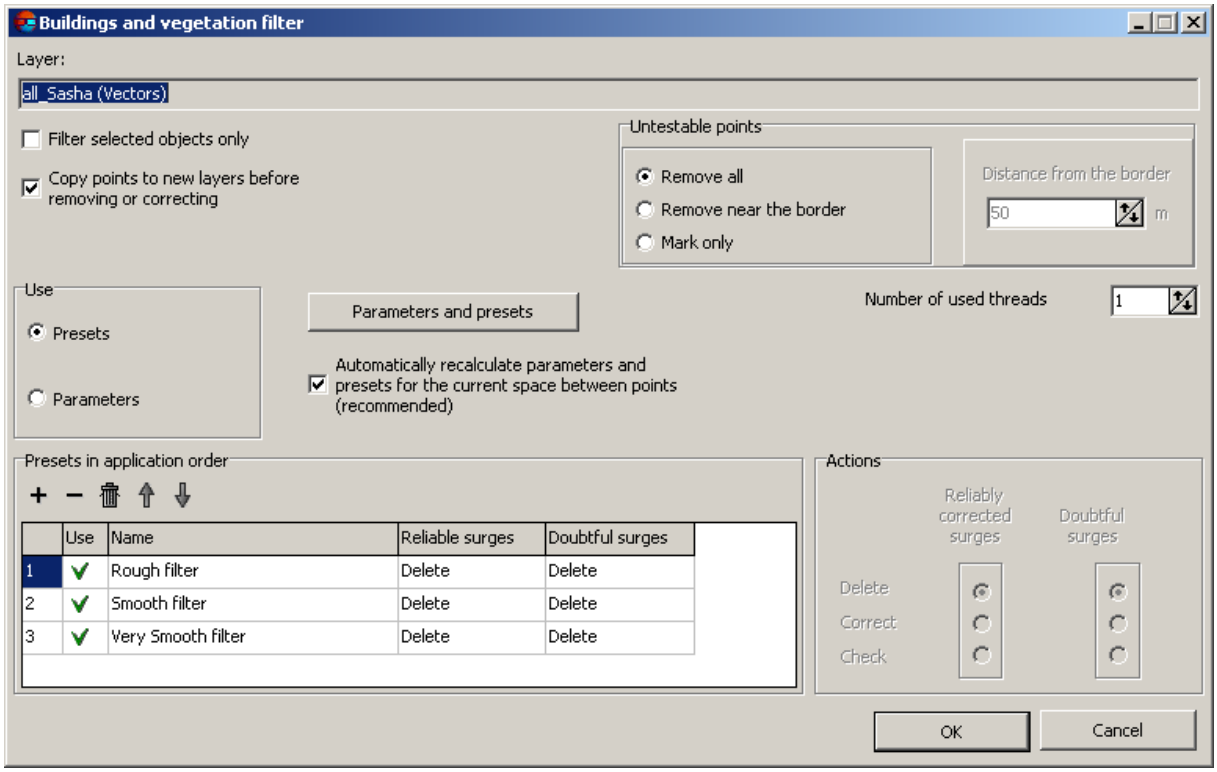
It is possible to create new custom presets. The different combinations of presets and sequence of their application allows producing optimum filtration results for the given area, in order to obtain points laying strictly on terrain, which in turn can be used to create quality DEM (see chapter Digital Elevation Model(DEM).

The following actions are available for detected erroneous points:

- Removal of points from the source layer;
- Correction of found surges (height adjustment) in the source layer of points;
- Checking points (saving found points to new layers for further analysis without editing the source layer).

To perform filtering of objects on the surface, do the following:

1. Load a vector layer with point objects (layer must be active). It is recommended to create a backup copy of layer data.
2. Use the **Vectors | Filter | Buildings and vegetation filter** menu command. The following window is opened:



Buildings and vegetation filter window

Name of the active vector layer of point objects is shown in the **Layer** field.

Filter selected objects only option allows applying filtering only to selected objects, for example, if it is required to reprocess only a certain area after the filtration is mainly complete for all points. In this case it is recommended to uncheck the **Remove untestable points located from the boundary** option in the **Untestable points** field.

Copy points to new layers section is used for saving the filtered out, corrected or untestable points in the new vector layers for analysis of found surges. Besides, these layers can be used to restore the base points layer (by merging layers), in case there is no backup copy of the source points layer. Some points may appear untestable during filtering. Usually these are points, neighborhood of which contains not enough other points – on the layer boundary or in areas where the correlator has found fewer points (fields, forests, water, etc).
Untestable points field allows to specify the actions with found untestable points:

- **Remove all** – allows to remove all the found untestable points;
- **Remove near the border** – allows to enter the distance from the edge of layer (not stereo pair, but all loaded layer data), which defines the untestable points to be removed;
- **Mark only** – allows to save untestable points into the separate layer.

Number of used threads parameter allows to use the multiple CPU's cores for implementation of distribution processing. Specify the number of CPU's cores in the input field to distribute into the threads.

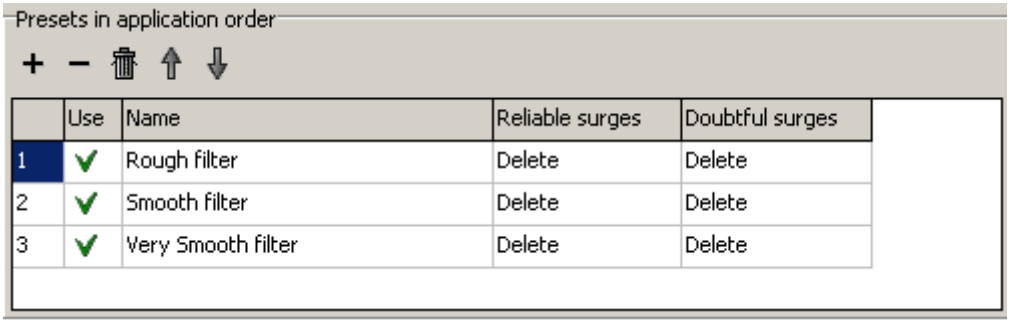
The **Use** section allows implementing two approaches to the objects filtration and contains the following options:

- **Presets** – allows using the existing pre-defined or user-created presets to filter in any combination and sequence of their application. Creating and configuring the parameter presets is implemented in the Parameters and presets window (called by pressing the appropriate button). By default, the list contains three pre-defined parameters presets by default - the **Basic filter**, the **Additional filter**, the **Detailed filter**. Filter scenario is created with the help of the **Presets in application order** table, in which the presets list is formed and actions for found errors (**Delete**, **Correct** or **Check**) are defined.
- **Parameters** – allows implementing only one pass with the specified parameters (in the Parameters and presets window). At that it is possible to use parameters of any existing preset in the list or set custom ones. After confirming parameters it is just necessary to choose actions for surges, which are found during filtration, in the **Actions** section: Delete, Correct or Check surges.

The **Parameters and presets** button opens the **Filter's parameters and presets** window for setting parameters, creating presets as well as importing / exporting the presets (see chapter Setting up parameters of filter).

The **Automatically recalculate parameters and presets for the current space between points (recommended)** option is used to automatically calculate the mean distance between points of the active layer and recalculating current parameters in accordance with the calculated step. It may be applied to existing presets.

The **Presets in application order** table is available if the **Presets** option in the **Use** section is selected. It is used to select preinstalled or custom presets (created in the Parameters and presets window), determine its application sequence (quantity of passes) and actions for error points detected during the filtering for each pass/preset. I.e. the table allows defining the filter scenario.



The table contains the following fields:

- Pass number;
- **Use** – it is possible to exclude the preset (empty field) from the list or include (the green check mark) with mouse click;
- **Name** – name of the preset, which is selected from the drop-down list of pre-defined or custom presets
- **Reliable surges** – this field defines actions for the reliably detected surges; the action is selected from the drop-down list (**Delete**, **Correct** or **Check** the points). When either **Delete** or **Correct** action is chosen, the active points layer is edited; copies of deleted or corrected points may be saved into new layers (if the corresponding option in **Copy points to new layers before removing or correcting** section is checked). If the **Check** action is chosen, the source points layer is left unmodified, and found surges are copied to a new layer *Surge points*. Surge points are treated as reliable when they satisfy all filtering criteria.
- **Doubtful surges** – this field defines actions for doubtful (uncertain) surges. Available actions are similar to those in the **Reliable surges** field. Points are treated as doubtful surges when there is not enough data in their neighborhood to make unambiguous analysis.

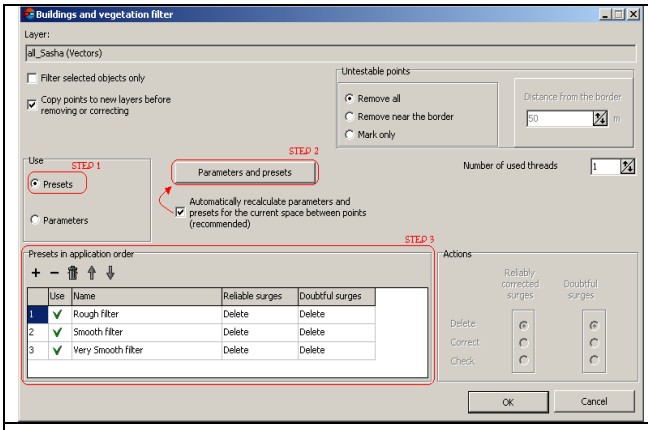
A toolbar on top contains the following buttons:

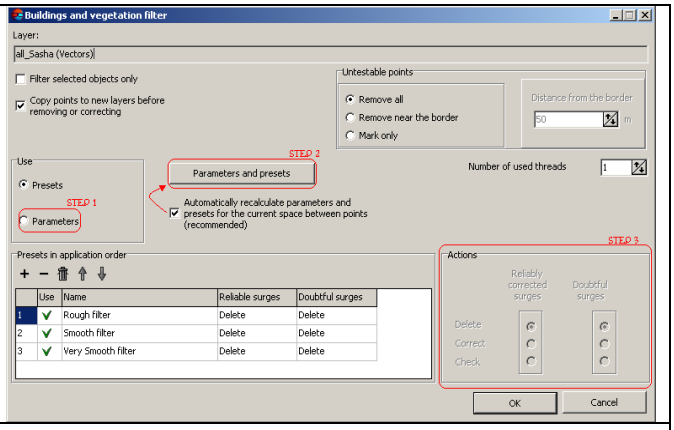
- – add a preset to the list (add a new row for the new preset);
- – remove selected preset from the list;
- – clear the list;
- – move the selected preset upwards;
- – move the selected preset downwards.

All these actions operate only on links to presets, and do not affect the presets themselves. In particular, clearing the list does not delete the presets.

The **Actions** section is available when the **Parameters** option in the **Use** section is checked; it also allows defining actions for reliably detected and doubtful points, found as a result of one-pass filtering with the given parameters.

Workflow for multi-pass filtration using combination of presets:	*Workflow for single-pass filtration with given parameters:*





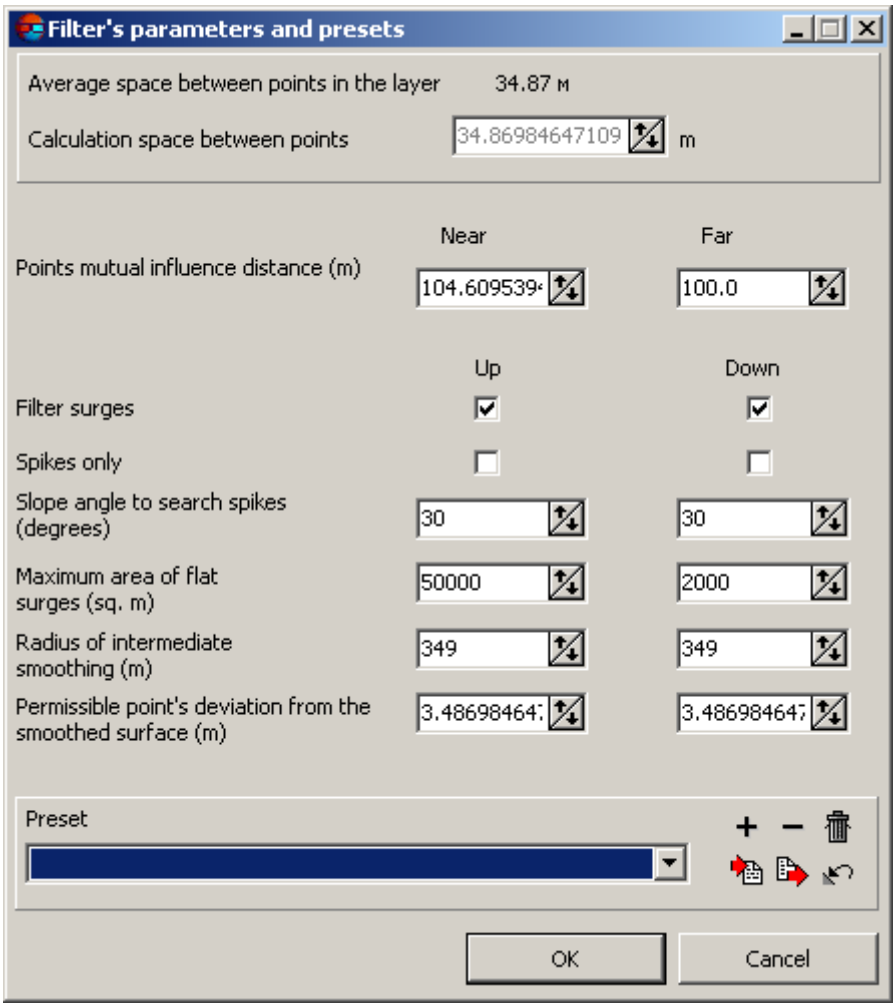
- Choose the **Presets** option in *step 1*.
- In step 2, press the **Parameters and presets** button to open the corresponding window for viewing existing or creating new presets. Existing preset parameters cannot be edited manually, but it is possible to automatically recalculate settings of all presets based on current mean distance between points (when **Automatically recalculate parameters…** option is on).
- Determine the filter scenario in *step 3* – the combination of presets, order of their application and actions for found surges.

- Choose the **Parameters** option in *step 1*.
- Press the **Parameters and presets** button in *step 2* for settings the parameters for the single pass. In this case it is possible to set/edit any parameter, as well as determine parameters automatically (when **Automatically recalculate parameters…** option is on). See chapter Setting up parameters of filter.
- Choose action for found surges in *step 3.*

See also chapter Recommendations on using the filter.

6.3.2.9.1. Setting up parameters of filter

The **Filter's parameters and presets** window is opened by pressing the **Parameters and presets** button in the **Buildings and vegetation filter** window. It is used for defining parameters and creating, exporting and importing parameter presets.



Parameters and presets

When filtering by parameters is chosen (option **Use / Parameters** in the **Buildings and vegetation filter** window), the parameters set in this window are used. If the **Automatically recalculate parameters and presets for the current space between points** option is on, the parameters appear already recalculated when the window is displayed. Then, it is possible to edit any parameters.

When filtering by presets is chosen (option **Use / Presets** in the **Buildings and vegetation filter** window), the parameters saved in presets are used. If the **Automatically recalculate parameters and presets for the current space between points** option is on, the recalculated parameters are used (the presets themselves are still not modified). I.e. it is not possible to edit the existing preset parameters, but the preset may be used as a base to create another preset.

The **Filter's parameters and presets** window contains the following filtration parameters:

- **Average space between points in the layer**: displays mean distance between points in the active layer.
- **Calculation space between points**: the field can be edited if the **Automatically recalculate parameters and presets for the current space between points** option in the **Buildings and vegetation filter** window is off; when this option is on, the value of this field is equal to the mean space between points (rounded to integer value). When the **Calculation space between points** is changed, the rest of parameters are recalculated automatically.

- **Points mutual influence distance (m)** - circle radius, in which area the values pickets heights will be analyzed for errors. It is recommended to set the value of 3-5 the average distances between the pickets for the near distance (the minimum circle radius).
- **Filter surges (up/down)**: if either option is on, the filtering is performed. If only *up* option is on, the program ignores surges directed downwards, and vice versa.
- **Spikes only (up/down)**: if the option is on, the points are filtered by two criteria only: slope angle to search spikes and points mutual influence distance (see below). The rest of parameters are not used (and are disabled).
- **Slope angle to search spikes** – this angle is counted off from the horizontal plane.
- **Maximum area of flat surges** – the groups of points forming smooth surfaces and displaced from some mean adjacent points plane are detected given the area of such surfaces is less than this parameter. Usually, these are points on flat roofs of buildings. Surfaces with larger areas are not treated as surges.
- **Radius of intermediate smoothing** – radius of the sphere defining the surface smoothing.
- **Permissible point's deviation from the smoothed surface** – defines a criterion, according to which points, having distance larger than the given value from the smoothed surface, are filtered.

The **Preset** list contains a drop-down list of all presets (pre-defined and custom). Any preset is a combination of defined parameter settings, which are recalculated when the **Calculation space between points** parameter is changed.

The following toolbar buttons are used for managing the presets:

– add a new preset to the list. The button should be pressed *after* the parameters for a new preset are ready, to open a window for naming the preset. After the name is specified, the preset is created from current parameters, and it's name appears in the presets list.

– delete the selected preset from the list

– clear the presets list

– export all presets from the list into a file

– import presets from a file into the list

– reset to the initial state – delete all custom presets and reset the pre-defined presets to their default settings.

6.3.2.9.2. Recommendations on using the filter

The following should be taken into account when setting up the filter:

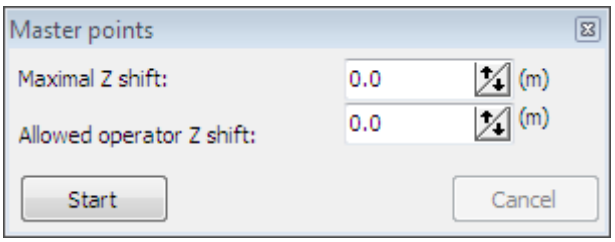
1. It is preferable to use stagewise filtering (several passes with different parameter presets).
2. The presets should be formed in the order of increase of the following parameters: points mutual influence distance, radius of intermediate smoothing, and permissible point's deviation from the smoothed surface.
3. The radius of intermediate smoothing directly describes the terrain feature, thus it should not be set too large (more than 1500 m)
4. It is recommended to analyze beforehand the maximum area of flat surges in the area of interest (e.g. measure in stereopair window) and compare it to the value in preset parameters. In case the latter is less than measured area, the parameter should be increased.

5. Points mutual influence distance should not be set more than 20 times larger than the mean distance between points. It may lead to significant filtering time increase.

6.3.2.10. *Considering operator personal difference*

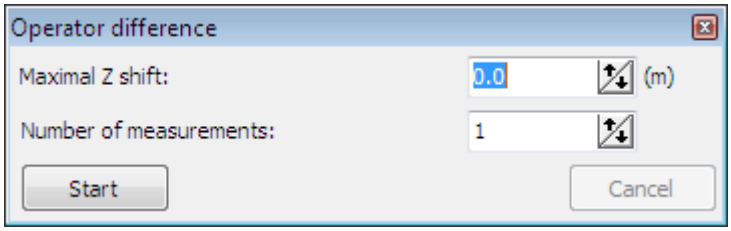
In practice, heights measurement accuracy in stereo mode depends on individual features of operator's vision. It is expressed in the fact that, along with random errors, the result of each operator work contains some permanent shift by Z coordinate that is specific just for this individual. Such effect is called "operator personal difference" and **PHOTOMOD** system provides it's considering and eliminating.

On the first step a set of pickets is created, which has its Z coordinate determined with the greatest accuracy. To do this, you must create or download from the resources set some pickets with quite accurate location on the surface. Then choose menu item **Vectors | Operator bias | Master points** to open the **Master points** window.



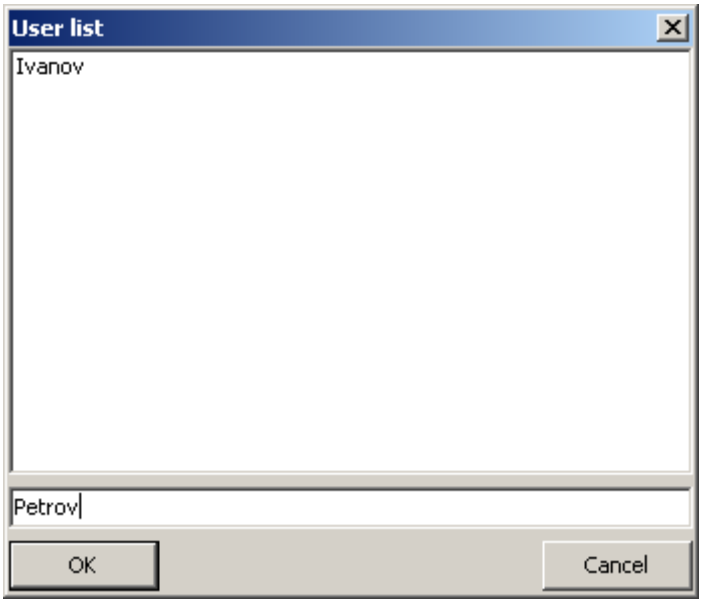
In this window a maximum shift on Z coordinate is set so, that it includes a guaranteed "ideal" position of every picket of the layer. Acceptable error is calculated as maximum difference between real and "ideal" location of each picket. Push the **Start** button. After that pickets no longer appear, and marker starts being set at coordinates of each picket, while height is being increased by random offset not exceeding **Maximal Z shift** parameter. An experienced operator should move the marker in height to the most accurate location (according to his point of view) and save this position using the **Enter** key (marker's XY coordinates are fixed then and can not be changed). Totally two passes for all pickets are made. Then passes are repeated only for those pickets which have measurement errors exceeding acceptable. After all the pickets have been measured with sufficient accuracy, the operator is proposed to save them as a vector resource of **PHOTOMOD** system.

On the second phase calibration of each operator by set of master points is performed. To do it you need to download master points and choose menu item **Vectors | Operator bias | Operator difference measurement**. You will see appropriate window.



In this window the maximum shift on Z coordinate is set in such a way that it is guaranteed to overlap a constant error of operator being calibrated. Number of measurements for each point can vary from 1 to 1000. Then push the **Start** button. After that pickets no longer appear, and marker starts being set at coordinates of each picket, while height is being increased by random offset not exceeding **Maximal Z shift** parameter. Operator being calibrated should move the marker in height to the most accurate location (according to his point of view) and save this position using the **Enter** key (marker's XY coordinates are fixed then and can not be changed). Upon completion of all the passes a message appears:

"**Operator difference: <number>**". After that a window appears with a list of operators, and you should enter a personal difference of operator being calibrated with his name specified.



After entering name and clicking **OK**, operator personal difference is stored in the database of operators.

The third phase includes accounting of personal difference in stereo mode process. To do so, select menu command **Vectors | Operator bias | Select operator difference**. This will display a window with a list of calibrated operators. After choosing the operator, his personal difference appears in the Status bar in red color.

To cancel the accounting of personal difference, double-click the line with operator personal difference in the Status bar. In window opened disable the option **Adjustment in current zoom pixels** and click **OK**. Operator personal difference will be equal to 0 after that.

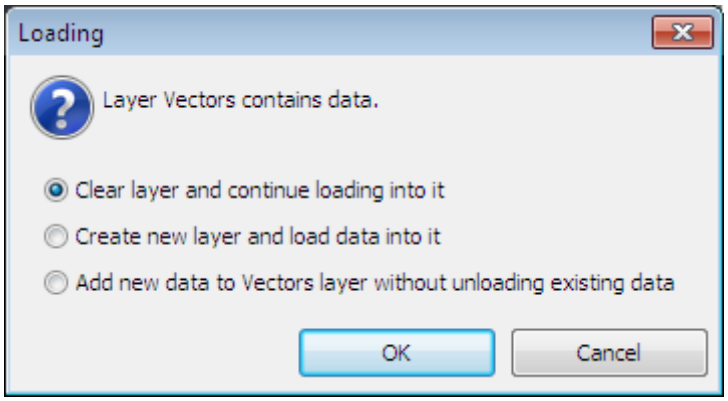
6.3.3. Using triangulation points

Triangulation points acquired on aerial triangulation stage can be also used for DTM creation (see **Aerial triangulation** User Guide), i.e. used as a vector layer.

Select menu command **Orientation | Load as vector objects** to display tie and ground control points on a block scheme. After that vector layer **Triangulation points (Vectors)** appears in **Layer Manager** and you can select it as one of base layers for TIN creation. See the chapter Building TIN.
Triangulation points are loaded as conventional vector objects without classifier with their names as labels. The layer may be saved to .x-data resource by standard means for working with vector layers.

6.3.4. Opening of base layers

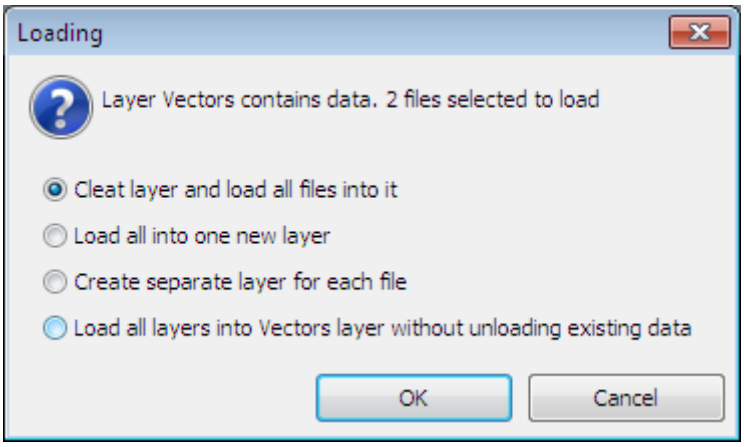
In order to open base vector layers, needed for TIN creation, use menu command **Vectors | Load**. Select a file with vector data in profile resources (file with x-data extension). Push the **Open** button. If some vector layers are already opened, the **Loading** window appears, that is used to specify the layer where to open the vector objects.


Loading
Layer Vectors contains data.
Clear layer and continue loading into it
Create new layer and load data into it
Add new data to Vectors layer without unloading existing data
OK
Cancel

There are the following options for data loading:

- **Clear layer and continue loading into it** – active vector layer is cleared from data and objects of selected file are loaded to it instead. At that the layer is renamed according to a name of loaded file. If no one of previously opened vector layers is active during loading, the last opened vector layer is suggested to use.
- **Create new layer and load data into it** – for objects of selected file is created a new layer with name, which is organized in the following way: <selected file name.x-data (Vectors(N))>, where N – is a number of regular vector file.
- **Add new data to Vectors layer without unloading existing data** – does not clear an active vector layer number N, but adds there vector objects from selected layer. If no one of previously opened vector layers is active during loading, the last opened vector layer is suggested to use.

When loading several files with vector objects at once, the program suggests to load all vector objects of all vector files into one new layer or to create separate layers for each vector file.


Loading
Layer Vectors contains data. 2 files selected to load
Cleat layer and load all files into it
Load all into one new layer
Create separate layer for each file
Load all layers into Vectors layer without unloading existing data
OK
Cancel

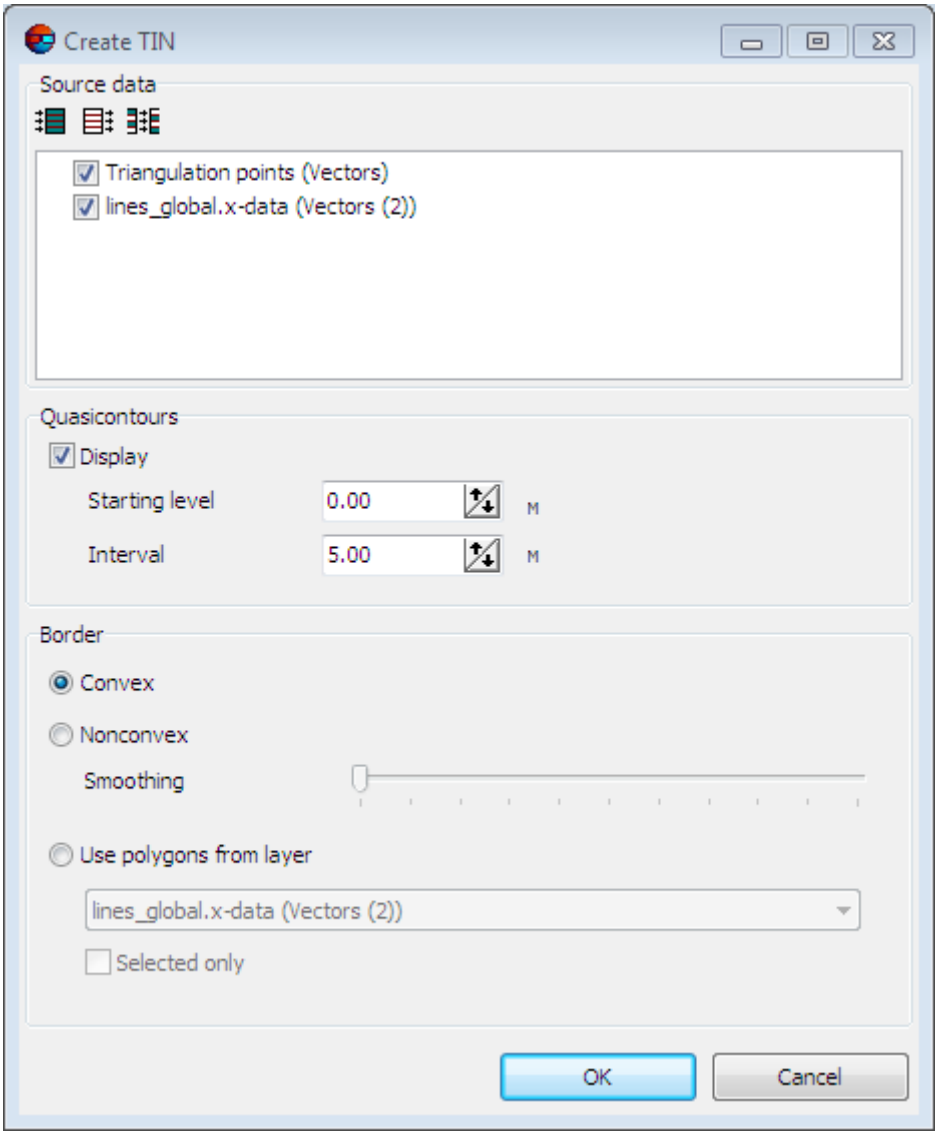
6.4. *Building TIN*

After forming and opening of all base vector layers you can proceed to TIN creation. See the chapter Preparing of base layers for TIN creation. At that, you can specify in advance boundaries of future TIN. See the chapter Building of TIN boundaries.

For TIN creation use menu command **TIN | Create**, that opens the Create TIN window.

6.4.1. **Create TIN** window

Create TIN window is opened by menu command **TIN | Create** and contains the following parameters:

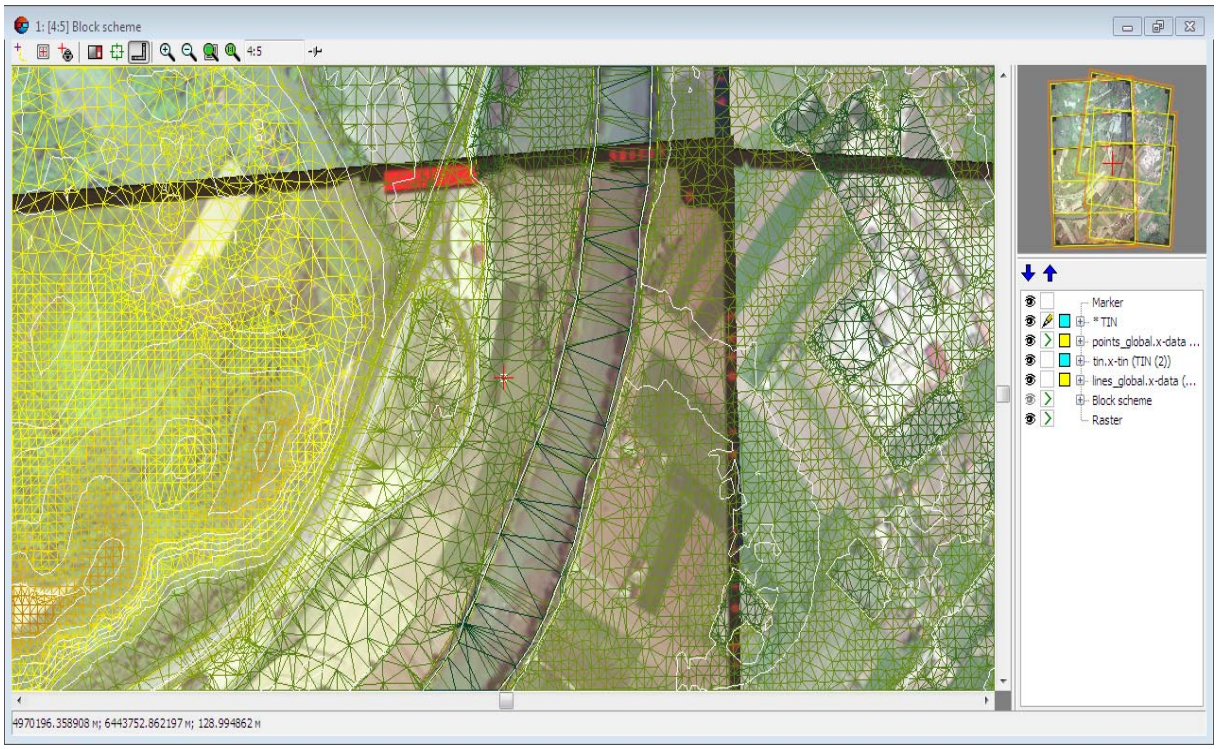


The **Source data** section contains a list of all opened vector objects (all of them are selected by default). Select base layers that will be used for TIN creation and dynamic rebuilding. For base layers selection you can also use **Select all** and **Unselect all** tools. See also the chapter Preparing of base layers for TIN creation.

Quasi-contours – are isolines that are created dynamically over sides of TIN triangles. Quasi-contours are integral part of TIN and are also rebuild “on-the-fly” along with TIN during base layers editing. Quasi-contours are used as a rule for additional quality control of TIN creation and blunders detection. If it is needed, select the **Display** option and specify the starting level and step of quasi-contours in meters.

The **Border** section allows:
– to specify the type of TIN border – convex or non-convex. In this case TIN covers all vector objects of base layers.
– to create TIN using specified border – polygon(s) of defined vector layer. In this case TIN is calculated just in boundaries of specified polygons. TIN created by the polygons has convex border. See the chapter Building of TIN boundaries.

Click **OK**. If there is TIN in **Manager**, the **Loading** window is opened containing options of loading – clear current layer with TIN and load there a new one or load new TIN into new layer.



TIN with quasi-contours

The following capabilities for working with TIN are available:

- various TIN display modes for visual analysis (see chapter Viewing TIN).
- calculation of TIN surface area (3D and projection on XY plane, see Calculating TIN area
- filtering TIN and TIN topology control (see chapter TIN editing).
- TIN accuracy control (see chapter TIN control against triangulation points).

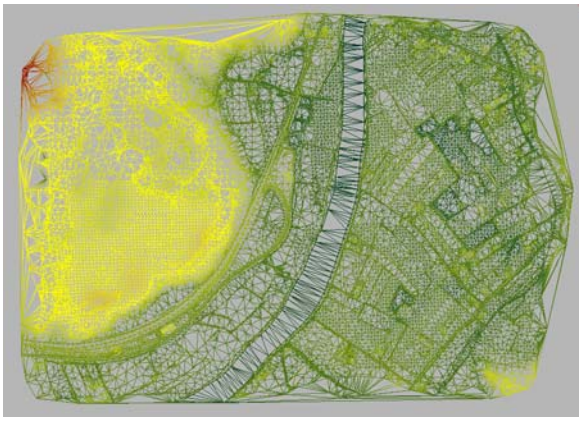
6.4.2. Building of TIN boundaries

TIN may have convex or non-convex border. It is recommended to create TIN with convex border if the initial vector objects are distributed so, that a part of a model is not “covered” by them (for example, rivers or lakes on large scale images).

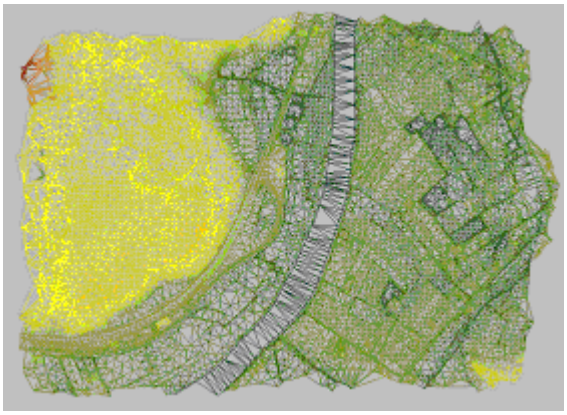
During TIN creation all vector objects of all base layers are built-in to the TIN. However you can define explicitly the area of TIN creation – by specifying the TIN border. To do that you should create polygons-areas on particular vector layer. You can draw the polygons manually, but also obtain them by automatic calculation of borders using specific parameters. Created polygons may be edited and used as TIN borders.

Creation of TIN without pre-defined border

In **Create TIN** window in *Border* section unselect the option **Use polygons from layers** and select only type of TIN border – convex or non-convex (with some smoothing level if necessary).



Example of TIN with convex border



Example of TIN with non-convex border

Creation of TIN with pre-defined border

It is possible to use two ways – create borders manually or automatically:

1. Draw polygon(s) on a separate vector layer and specify it as a TIN border. For that in Create TIN window enable the **Use polygons from layers** option and select a layer with created border. You can also use only selected objects of the layer (by enabling the appropriate checkbox). Push **OK**. TIN will be created within specified border. The TIN border will be convex.

2. You can also calculate and create the border automatically. The border is calculated considering vector layers. For that, use menu command **TIN | Compute border**. The **Compute TIN border** window is opened. Specify base vector layers, for which TIN border should be calculated. Select border type:

- Convex
- Non-convex (with smoothing level specified)
- Non-convex with splitting into regions. In this case, the program analyses vector objects distribution and split them into regions if there are gaps.

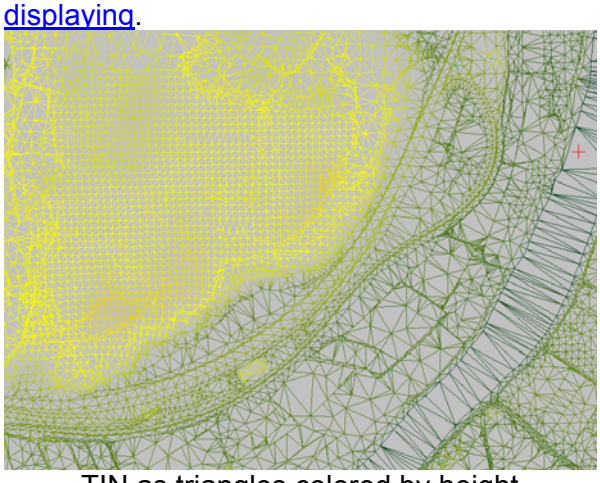
Push **OK**. The *TIN border* layer appears in **Manager** and contains created border that is vector polygon (polygons).
For TIN creation using pre-build border, in Create TIN window enable the **Use polygons from layers** option and select the *TIN border* layer.

TIN border calculation

The figure above shows convex border by green color, non-convex – by blue (the same for all vector objects of base layers), and by red – two split borders-regions. Vector objects of base layers are displayed in grey color.

6.4.3. Viewing TIN

The **TIN** tab of **Settings** window (opened by menu command **Service | Settings**) allows to setup parameters of TIN displaying in 2D window. The settings include displaying TIN as triangles colored by elevation, or solid fill. Also for enhanced visual experience, antialiased display may be enabled for TIN edges and quasi-contours. See chapter Settings of TIN displaying.



TIN as triangles colored by height

TIN as a solid filling

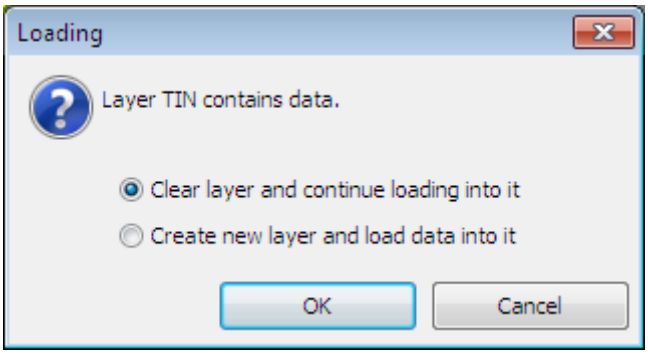
You can also view created TIN in 3D window using menu command **Window | 3D window**. See the chapter 3D Window.

6.4.4. Saving TIN

For TIN saving use menu command **TIN | Save** or **TIN | Save as**. TIN is saved in profile resources in file with .x-tin extension, which contains relative paths to vector resources – base layers, and also step of quasi-contours. If base layers are not saved, they are saved automatically during TIN saving.

6.4.5. Loading TIN

For TIN loading use menu command **TIN | Load**. Select TIN to be loaded (file with .x-tin extension) in profile resources and push the **Open** button. TIN is loaded along with base vector layers (if they are not opened yet).
You can load several TIN's placing them to separate layers. But you can also replace existing TIN during loading of next one.



6.4.6. Closing TIN

Prior to closing of selected TIN (using menu command **TIN | Close**) you will be suggested to save it to profile resources, and then to close its accompanying base layers too.
At closing of one of base layers, the TIN itself is also closed. Before closing it is suggested to save it to profile resources.

6.5. *TIN editing*

In **PHOTOMOD** system TIN does not exist without vectors, and it is impossible to edit it (i.e. its nodes and edges), since TIN contains just relative paths to base vector layers, and step of quasi-contours. You can change a net of triangles (TIN) only when editing the base – vector objects of base layers. TIN along with quasi-contours is re-built "on-the-fly" during editing of base layers. See the chapter Editing of vector objects.

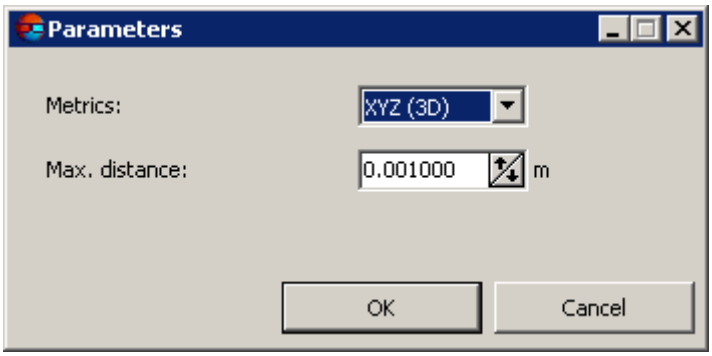
The following means for TIN editing are available in **PHOTOMOD**:

- **Filtering** – simplifying, peak filter and Z-range filter, which are applied to objects of base vector layers. See chapter Filtering.
- **Topology verification** – search for adjacent TIN nodes (i.e. points and line vertices of base vector layers) and topology correction. See chapter TIN topology verification.

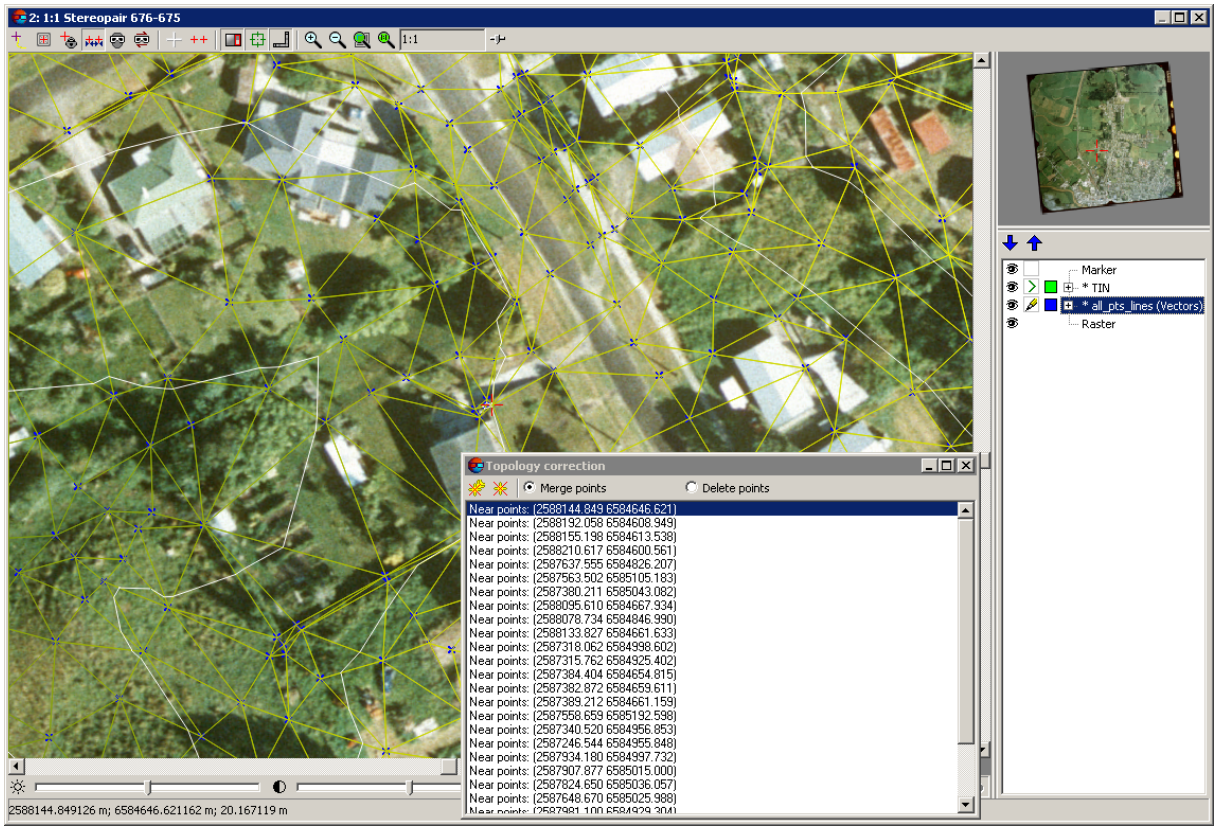
6.5.1. TIN topology verification

TIN topology verification consists in searching for adjacent TIN nodes, i.e. adjacent *vector points and polyline/polygon vertices* of base layers, which form the TIN triangle vertices.
Topology correction consists in deleting or aligning found adjacent points (i.e. editing points or vertices of vector objects). Aligning (joining) points consists in moving adjacent points to from a single point.

Execute the **TIN | Verify topology** menu command. The **Parameters** window is opened:



Select metrics to search for adjacent points either in plane – **XY (2D)** or considering the Z coordinate – **XYZ (3D)**. Set the maximum distance between points which are considered adjacent. The process started by pressing **OK** button results in the **Topology correction** window containing the list of found points, or a message informing that no topology errors have been found.
The window allows correcting the topology (erasing or aligning points) for all points in the list at once (button) or for the selected point only (button). When a point is selected in the list, the marker automatically moved to its position in 2D window. The issue may be fixed manually (see chapter Editing of vector objects). Consecutive review of each point in 2D window is recommended to avoid loss of valuable data acquired during stereovectorization.



TIN topology correction

6.5.2. Filtering

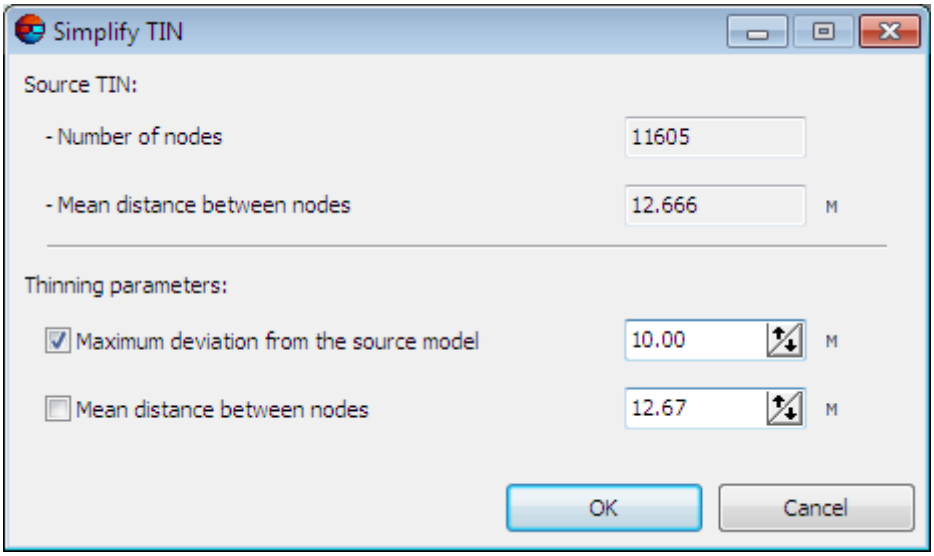
For TIN editing there are three filters available – simplifying, peak filtering and filter by Z-range. In case you apply simplifying and peak filtering you will delete “bad” pickets (point vector objects), which are TIN nodes. I.e. a deleting of TIN node means deleting of

corresponding vector point. If filtering by Z is applied the program deletes point objects and vertices of polylines/polygons, Z-coordinate of which lies outside of specified range. See also chapters Simplifying, Peak filtering and Filter by Z-range.

6.5.2.1. *Simplifying*

Simplifying of nodes of TIN triangles means filtering of point vector objects in base layers. To start simplifying select menu command **TIN | Filter | Simplify**. You will get the **Simplify TIN** dialogue opened, that contains initial TIN statistics and allows to setup the simplifying parameters.
The statistics includes information on the number of TIN nodes and mean distance between nodes (average length of edges). There are the following simplifying parameters:

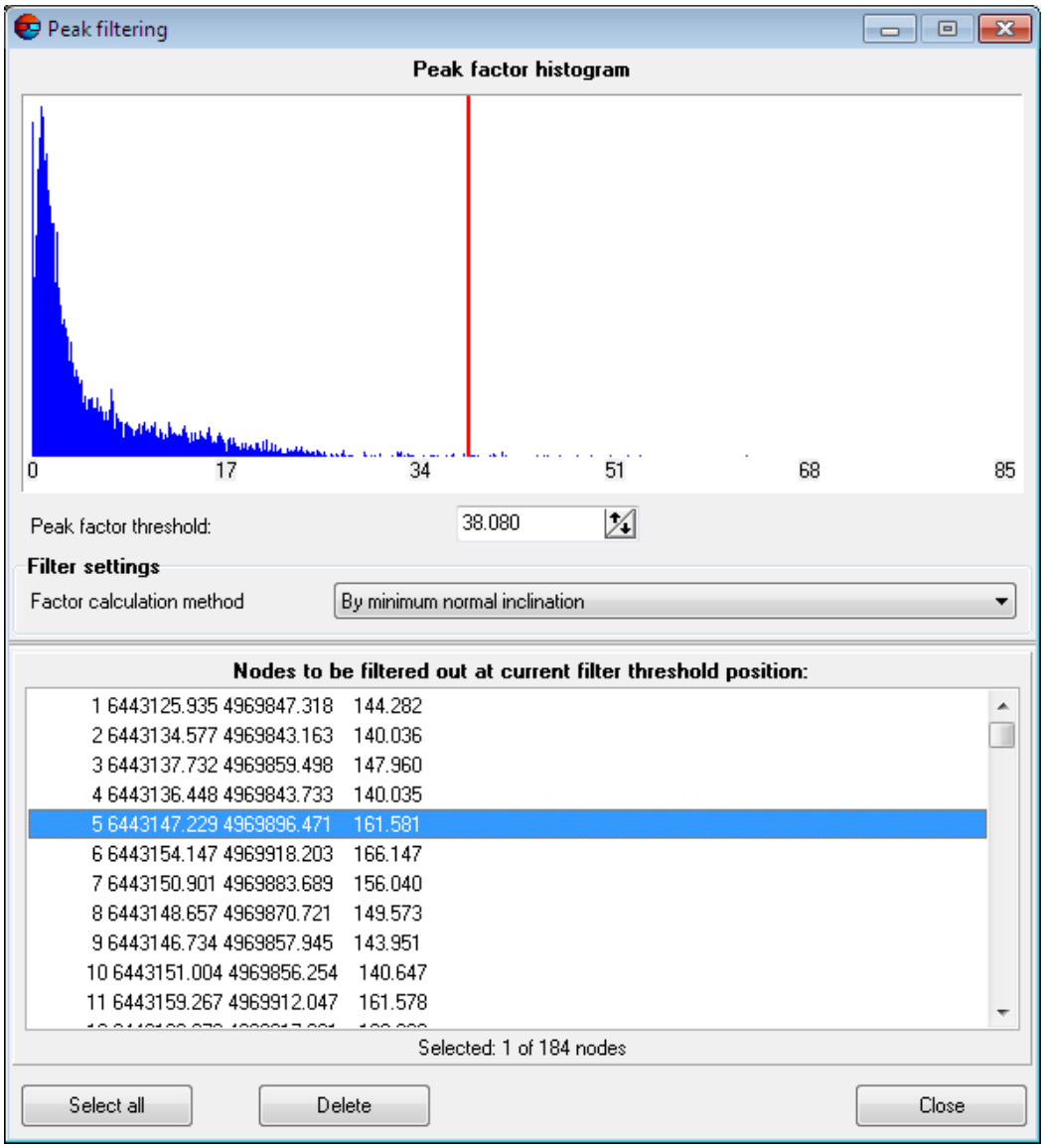
- **Maximum deviation from the source model** – maximal deviation of source model from resulting one by Z.
- **Mean distance between nodes** – used to setup the distance between nodes in meters.



Push **OK**. After simplifying is finished you will get a message about number of deleted nodes. Please remember, that deletion of TIN nodes means deletion of vector points from base layers. The filter works with all base layers as with one editable layer. To cancel filtering results use the **Undo last action** tool.

6.5.2.2. *Peak filtering*

Peak filtering operation (called using menu command **TIN | Filter | Peak filtering**) consists in searching for TIN nodes, which differ significantly from neighbor nodes by Z value. I.e. the system detects nodes of sharp local spikes. The filter is applied not for TIN nodes themselves, but directly to point objects (pickets) containing in base layers, that used for creating TIN nodes.
The **Peak filtering** window allows to delete a part of nodes, which form the most «peaked» pyramids.



Peak filtering

Relative distribution of TIN nodes with different peak factor is displayed on the histogram. The relative distribution of TIN nodes with different peak factor is displayed in the blue histogram. The node with peak factor equal to 0 is considered as “error-free”. The node of minimum reliability calculated in accordance with the selected method has peak factor value equal to 100. In order to filter TIN nodes move vertical red line by mouse or just enter the **Peak factor threshold** value from the keyboard.

The peak factor is calculated by one of the following methods (selected in the **Factor calculation method** drop-down list):

- **By minimum normal inclination** – peak factor is proportional to minimum angle value between the normal to triangle plane and the plane of Z = const for all triangles “around” the TIN node.
- **By maximum normal inclination** – peak factor is proportional to maximum angle value between the normal to triangle plane and the plane of Z = const for all triangles “around” the TIN node.
- **By mean normal inclination** – peak factor is proportional to mean angle value between the normal to triangle plane and the plane of Z = const for all triangles “around” the TIN node.
- **By sum normal projection** – peak factor is inversely proportional to the absolute value of projection of the vector sum of normal lines to triangles on the Z axis.

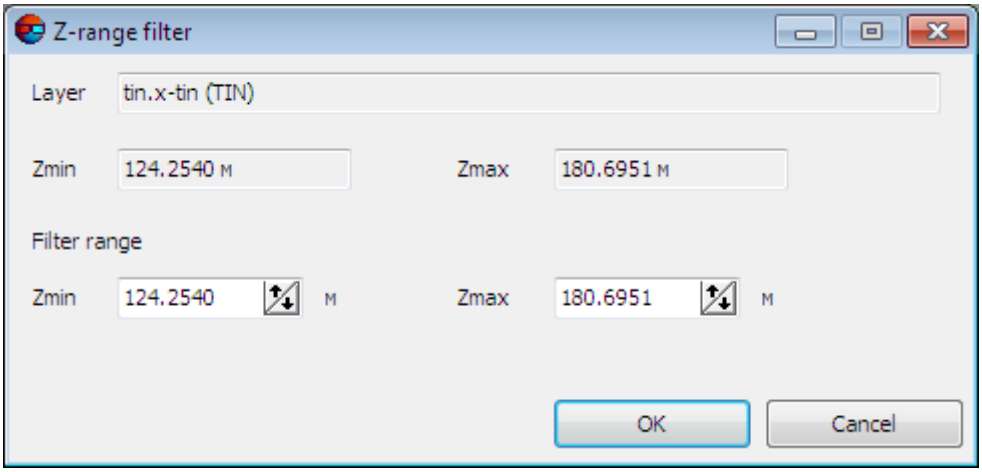
- **By sum normal absolute value** – peak factor is inversely proportional to the absolute value of the vector sum of normal lines to triangles on the Z axis.

After selecting the method for peaks and threshold calculating, push the **Refresh** button, after that in the lower window appears TIN nodes list according to these parameters. Mouse click on list record highlights the selected node in this list and in 2D window. The nodes group could be selected in the list by mouse along with pressed **Ctrl** or **Shift** keys. Nodes selected in such a way could be deleted using the **Delete** button.

Please remember, that peak filtering results in deletion of vector points from base layers and, therefore, in deletion of appropriate TIN nodes. The filter works with all base layers as if it is one editable layer. To cancel filtering results use the **Undo last action** tool.

6.5.2.3. *Filter by Z-range*

TIN editing with use of the filter by height means deleting of vector points, and vertices of polylines/polygons (corresponding to TIN nodes), Z-coordinate of which lies outside of specified range. To apply filtration of TIN by Z-range select menu command **TIN | Filter | Filter by Z-range**. The appropriate window will appear.

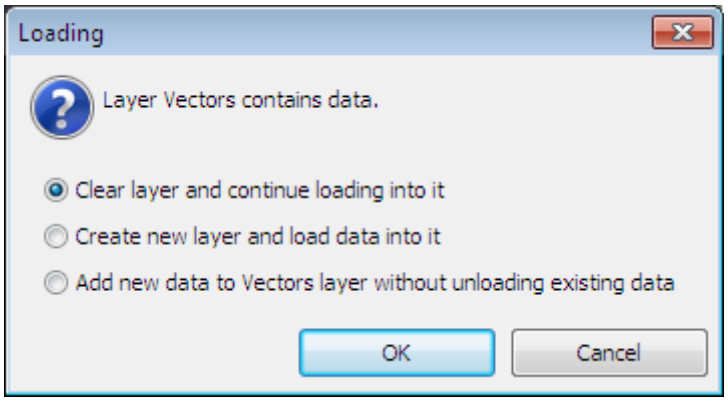


In the **Layer** panel you see the TIN to be filtered. **Zmin** and **Zmax** fields show TIN height range. Specify filter range by height in meters in appropriate section and push **OK**. To cancel filtering results use the **Undo last action** tool.

6.5.3. Rebuilding of TIN

If you need to rebuild TIN considering objects of new vector layer, you may use the following ways:

- If you choose to load “vectors” of new layer into base layer (by selecting option **Clear layer and continue loading into it** or **Add new data to Vectors layer without unloading existing data**) TIN is immediately re-build dynamically considering added vector objects.
- When you choose loading vector objects into new layer (by selecting option **Create new layer and load data into it**) you should build the TIN once again. For that perform menu command **TIN | Create** and select a new layer as base one.

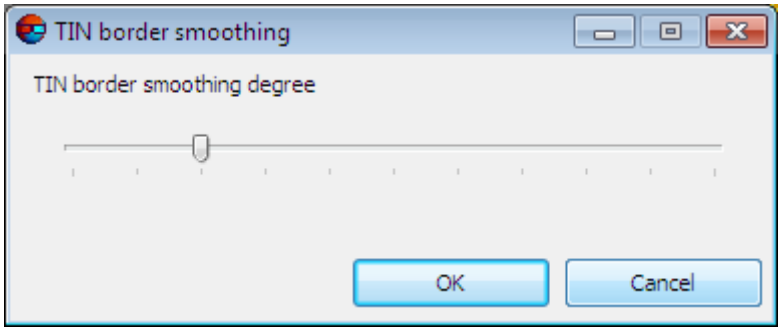

Loading
Layer Vectors contains data.
Clear layer and continue loading into it
Create new layer and load data into it
Add new data to Vectors layer without unloading existing data
OK
Cancel

Loading of new vector data

During closing of one of the base vector layers which is the TIN base, the TIN itself is closed too.

After TIN editing (i.e. editing of base layers objects) for final TIN re-creation perform menu command **TIN | Rebuild**.

Besides, the **Rebuild** command allows to change a level of smoothing of non-convex border of created TIN. Layer with TIN should be active (marked in Layer manager by or sign).

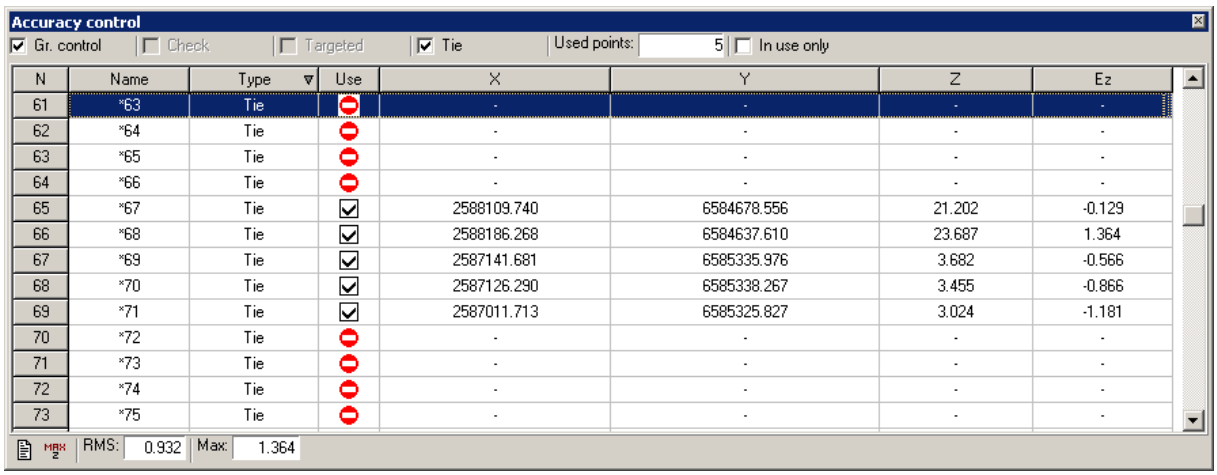


6.6. *TIN control against triangulation points*

If quasi-contours were built duting TIN creation, their analysis in stereo mode may reveal blunders in TIN model. See chapter TIN editing.

If triangulation points were not used as one of the base layers for TIN creation (see chapter Using triangulation points), it is feasible performing TIN checking against the triangulation points – analysis of TIN deviation by Z. In order to do it, execute the menu command **TIN | Check against adjustment points**. The **Accuracy control** window opens containing the list of all points (ground control/check/tie) obtained during the block triangulation stage (see the **Aerial triangulation** user manual).

© 2011



Accuracy control against triangulation points

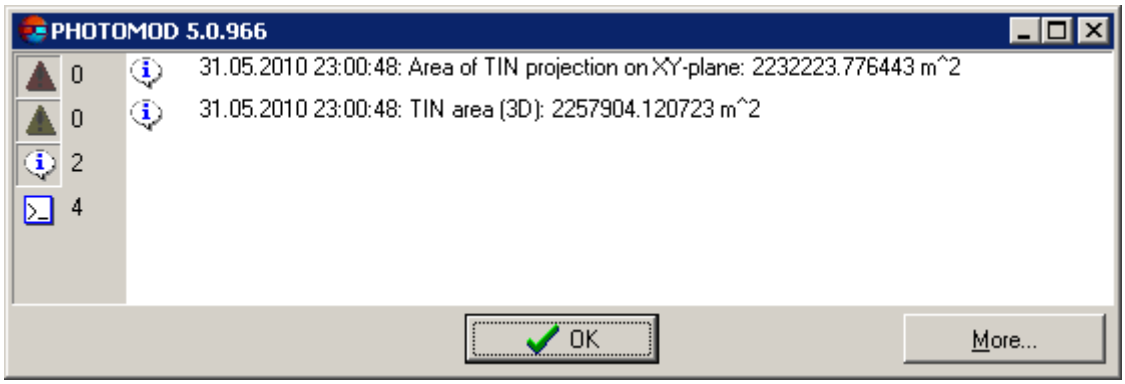
The top panel of the window is used to control visibility of different kinds of points in the list and allows displaying **ground control**, **check**, **tie** and/or **targeted** points, as well as used (falling within the TIN border, having the Z discrepancy computed) only points. Also the quantity of used points is displayed in the panel.
Each point in the list has the following data in the corresponding columns:

- point **Name** and **Type**;
- whether the point is **used** in TIN Z discrepancy control (- not used, - used);
- adjusted coordinates of the point (**X**,**Y** and **Z**);
- TIN discrepancy by Z (**Ez**).

Besides, the RMS and Maximum error by Z is displayed in the bottom panel.
The button is used to find and select in the list the point with maximum discrepancy.
The button is used to open and print the detailed accuracy report.
The **Name, Type, Use** and **Ez** columns may be sorted in ascending or descending order (by clicking the column title).

6.7. *Calculating TIN area*

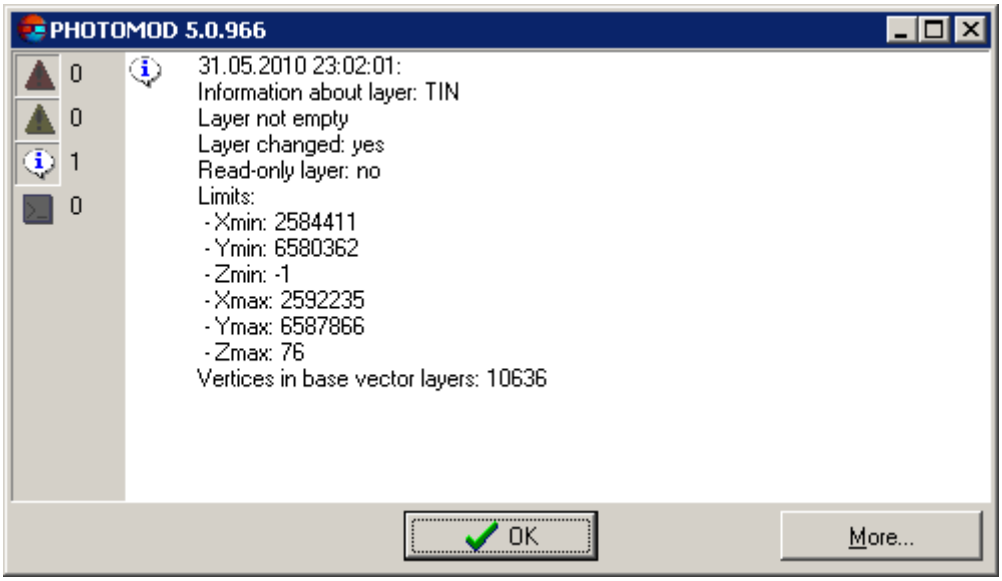
The system allows calculating area of all the TIN (surface and projection on XY plane), as well as area of TIN surface within a given polygon.
To do such calculations, execute the **TIN | Compute area** menu command. The system outputs areas of 3D TIN surface and TIN projection on XY plane in sq.m.



To compute area of the TIN surface within a polygon, select the polygon (create new or use existing in any vector layer) and execute menu command **TIN | Polygon area on surface**.

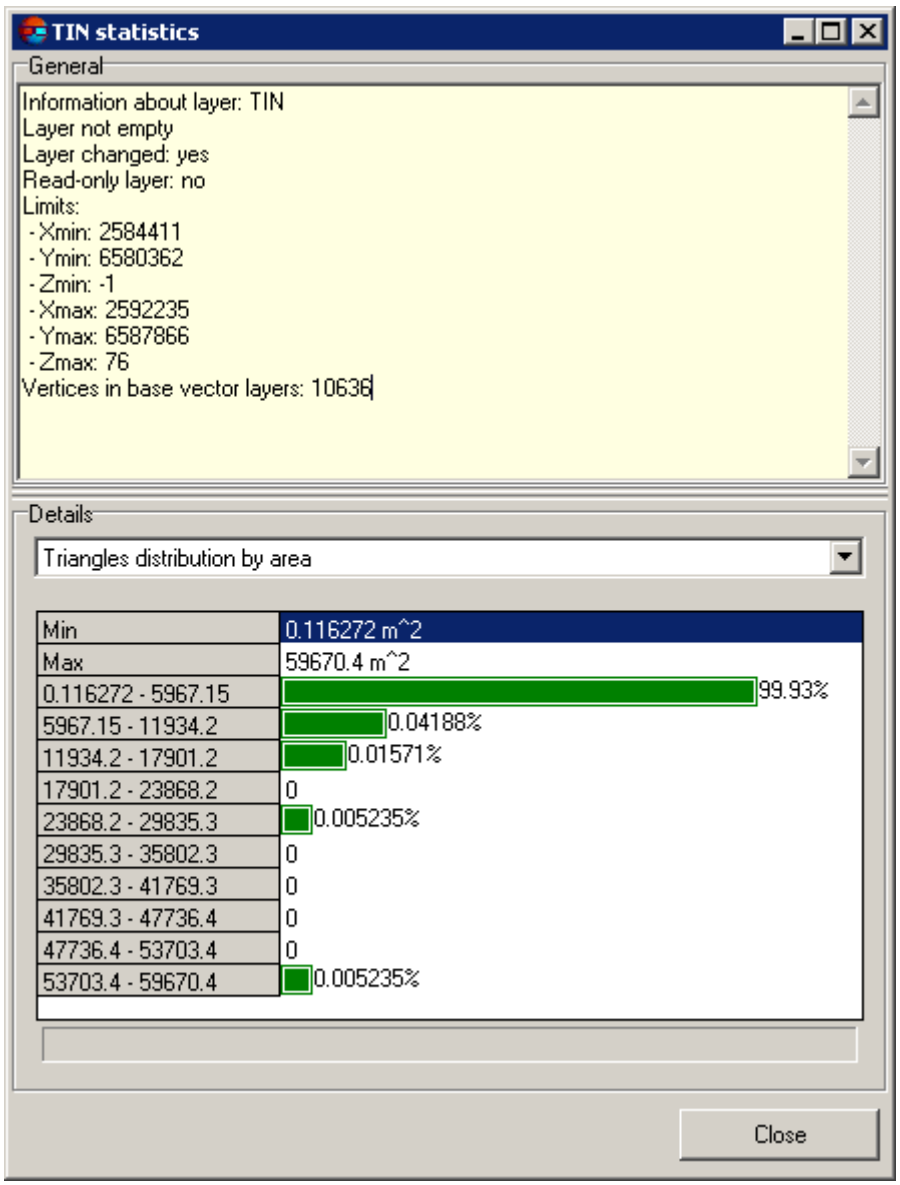
6.8. *Statistics*

Concise TIN summary can be retrieved by menu command **Edit | Active layer | Layer information**, or the **Information** command in the layer context menu on **Layer Manager** (see chapter Layer manager).



More details can be retrieved with **TIN | Statistics** menu command, which opens the **TIN statistics** window.
The upper section contains general layer data, the lower one – statistics on TIN triangles in bar chart form for the following parameters:

- Triangles distribution by area
- Nodes distribution along Z axis
- Triangles distribution by min. angle
- Triangles distribution by max. angle

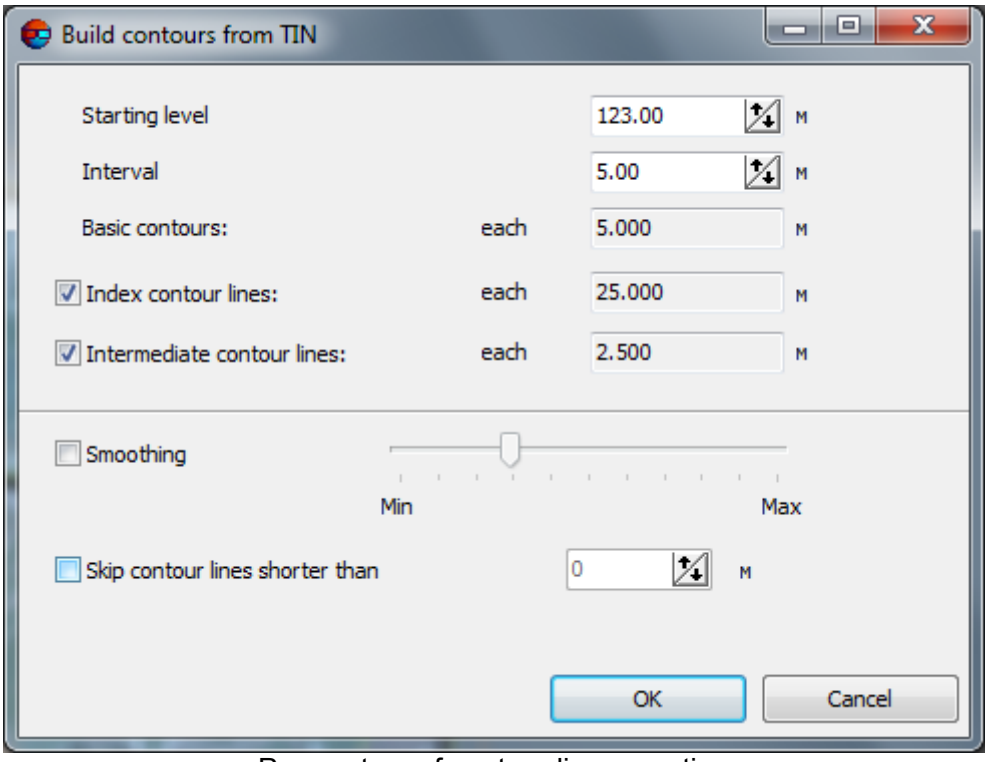


7. Contour lines

Source data for contour lines creation is relief model, represented as irregular network of triangles (TIN) or as regular digital elevation model (DEM). Apart from that, there is a capability of creating smooth-model contours from vector layers with given precision (see chapter Creating smooth model contours.

7.1. *Building contour lines from TIN*

In this case created TIN is used as a source data for contour lines building (see the chapter TIN).
To start contour lines creation using available TIN select menu command **Vectors | Build contours | From TIN**. The **Build contours from TIN** window is opened.



Parameters of contour lines creating

Here you can setup the following parameters:

- **Starting level** – Z value, which is used as a starting level for contours creation. Information on height range of TIN (Zmin, Zmax) you can see in statistics window, opened using context menu appeared after right click on the TIN layer in **Manager** or after performing menu command **Edit | Layer information**.
- **Interval** – contours interval in meters.
- **Basic contours** – step of basic contour lines (calculated automatically). If intermediate contour lines are disabled, step of basic contour lines equals to contour lines interval.
- **Index contour lines** – allows to create index contour lines with step equals to 5 steps of basic contour lines (calculated automatically).
- **Intermediate contour lines** – to create intermediate contour lines with step equals to step of basic contour lines (at that step of basic contour lines is recalculated and equals to 2 contours intervals).
- **Smoothing** – allows adjusting contour lines smoothing using a slider.
- **Skip contour lines shorter than (m)** – allows to exclude very short contours.

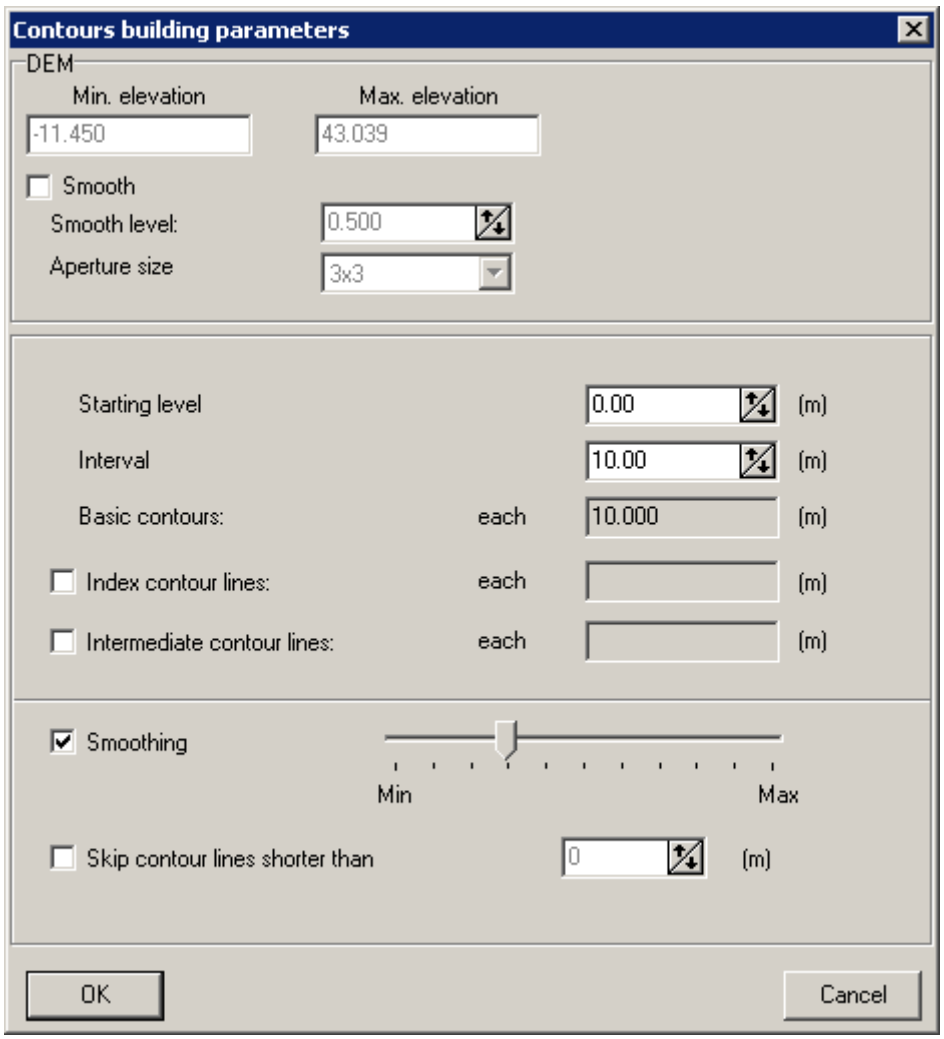
Push **OK**. The *Vectors* layer is created and operation of contour lines building is started.

To edit created contour lines use common tools of vector objects editing. See the chapter Editing of vector objects and Topology control.

7.2. *Building contour lines from DEM*

In this case digital elevation model (DEM) is used as a source data for contour lines building (see the chapter Digital Elevation Model (DEM)).

To start contour lines creation using available DEM select menu command **Vectors | Build contours | From DEM**. The **Contours building parameters** window is opened.



Parameters of contour lines creating by DEM

In **DEM** panel you can setup the following parameters:

- **Min. elevation** and **Max. elevation** fields show a range of DEM heights, that allows to estimate heights range for contours creation.
- **Smooth** – enables smoothing of initial DEM prior to contours building, considering the settings specified below. Algorithm of DEM smoothing in some node considers the information about its adjacent nodes, their number is defined by the **Aperture size** parameter. Smoothing level is defined by the **Smooth level** parameter, which can possess the value from 0 to 1, where the value 1 corresponds to maximal smoothing.

The **Contours** panel allows to setup contour lines features. There are the following obligatory parameters:

- **Starting level** – minimum Z value of contour, which used as a starting level for contours creation.
- **Interval** – contours interval in meters.

If necessary you can select **Minimal nodes number** – used to delete contour lines with a number of nodes less than the selected (the default value is 5).

Enable the option **Thick contours** to create thick contours according to the settings of this panel: starting level and the interval of thick contours (5 – by default).

In addition you can setup contours filtering parameters (by noise RMS) and contours smoothing parameters (by curvature threshold).

If the **Smooth** option is on, the smooth curves are created with given curvature.

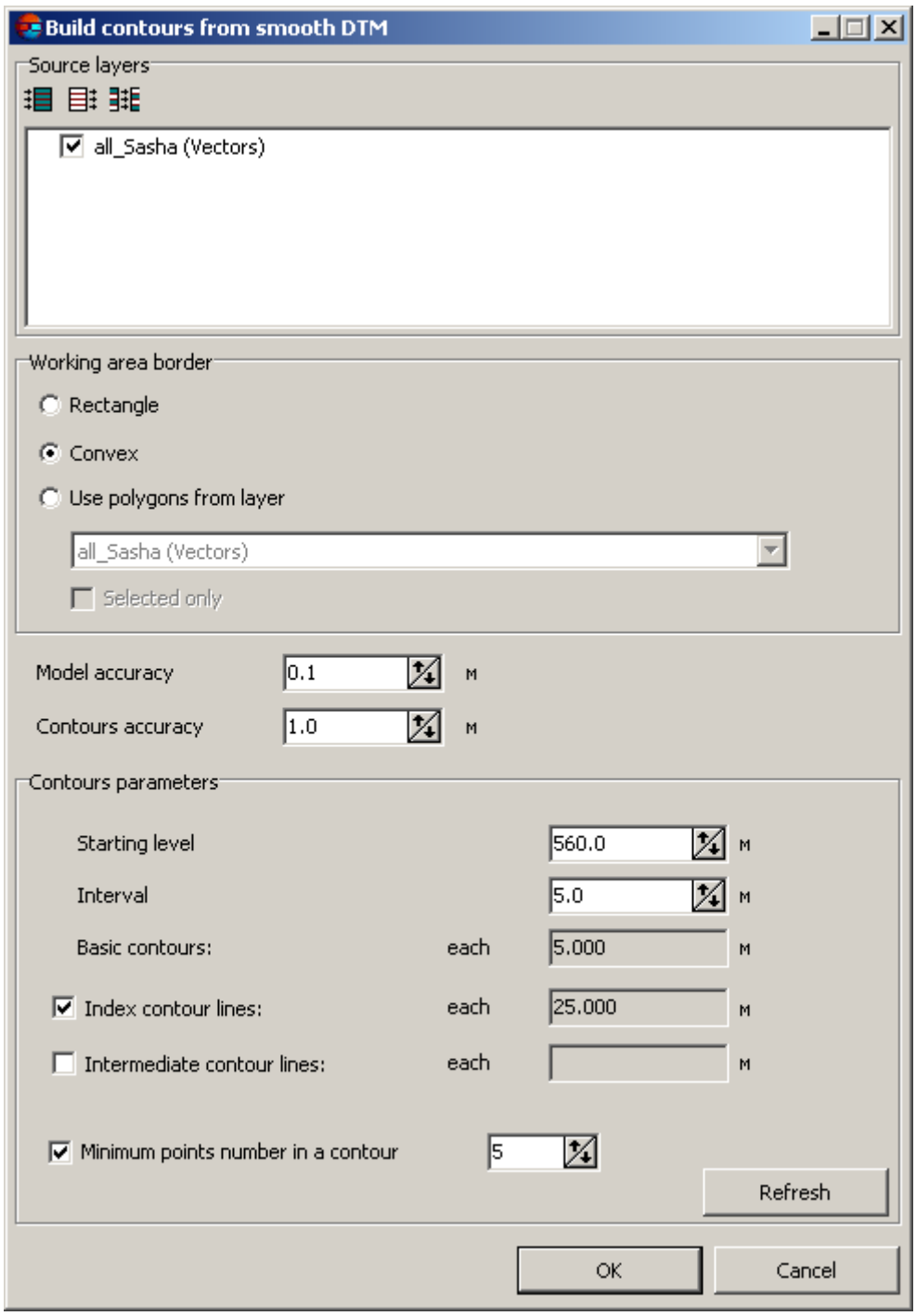
Push **OK**. The *Vectors* layer is created and operation of contour lines building is started according to your settings.

To edit created contour lines use common tools of vector objects editing (see the chapter Editing of vector objects, and Curves editing).

7.3. *Creating smooth model contours*

Creating smooth model contours allows creating contours as smooth curves with given precision. This type of contours creation guarantees absence of intersections, thus excluding the operation of contours intersection control.
The source data are any vector objects obtained as a result of stereovectorization. If needed, vector polygons may be defined to use them as borders of area of interest for creating the contours. See also chapter Vector objects creation.

To create the contours, load vector layers and execute the menu command **Vectors | Build contours | Smooth model**. The following window opens:



Select base layers to use for contours creation among the open layers. Define the area of interest:

- **Rectangle**
- **Convex**
- **Use polygons from layers** – select vector layer, all or selected polygons from which should be used as borders.

Define accuracy settings:
Model accuracy - maximum deviation of internal DEM along Z axis from base vectors. It should be set reasoning from the accuracy of vectors creation in stereomode. As a rule, this accuracy is about 1 pixel (GSD). The GSD can be found out e.g. in the grid properties window (**Grid | Properties**).

Contours accuracy – maximum deviation of contours along Z axis from internal DEM, which is used to create the smooth contours.
Next, specify the contours parameters (in project units): obligatory – starting level and interval of basic contours; if needed, enable creation of index and intermediate contour lines.

Minimum points number in a contour option allows to reject the contours, the number of nodes in which less then set value.

7.4. *Saving contour lines*

Contour lines are usual vector objects – polylines, created in separate *Vectors* layer. That is why for saving the layer with contour lines make it active in **Manager** and perform menu commands **Vectors | Save** to rewrite a current file or **Vectors | Save as** to input new file name and path to save them in profile resources.

7.5. *Loading contour lines*

Use menu command **Vectors | Load** to open contour lines available. In opened window with profile resources select vector file containing contour lines. Vector resources have x-data extension. You can open several resources with contour lines at the same time, by placing them into different layers.

There are the following options for contour lines loading:

- **Clear layer and continue loading into it** – active vector layer is cleared from data and selected contour lines are loaded to it instead. At that the layer is renamed according to a name of loaded file. If no one of previously opened vector layers is active during loading, the last opened vector layer is suggested to use.
- **Create new layer and load data into it** – for contour lines is created a new layer with name, which is organized in the following way: <selected file name.x-data (Vectors(N))>, where N – is a number of regular vector file.
- **Add new data to Vectors layer without unloading existing data** – adds to an active vector layer contour lines from selected layer. If no one of previously opened vector layers is active during loading, the last opened vector layer is suggested to use.

You can also open empty contour lines layer in **Manager** window using menu command **Vectors | Create layer** and then import there contours from exchange format using menu command **Vectors | Import**. Use **Vectors | Export** command to export created contour lines to format selected from a list.

7.6. *Operations on contour lines*

The created contours are in fact vector polylines consisting of straight segments (broken lines) or smoothed segments (curves), so they are subject to all editing operations on vector polylines and curves. See chapters Editing of vector objects, including Polyline/polygon editing and Curves editing.
Besides, the following special operations are available for contours:

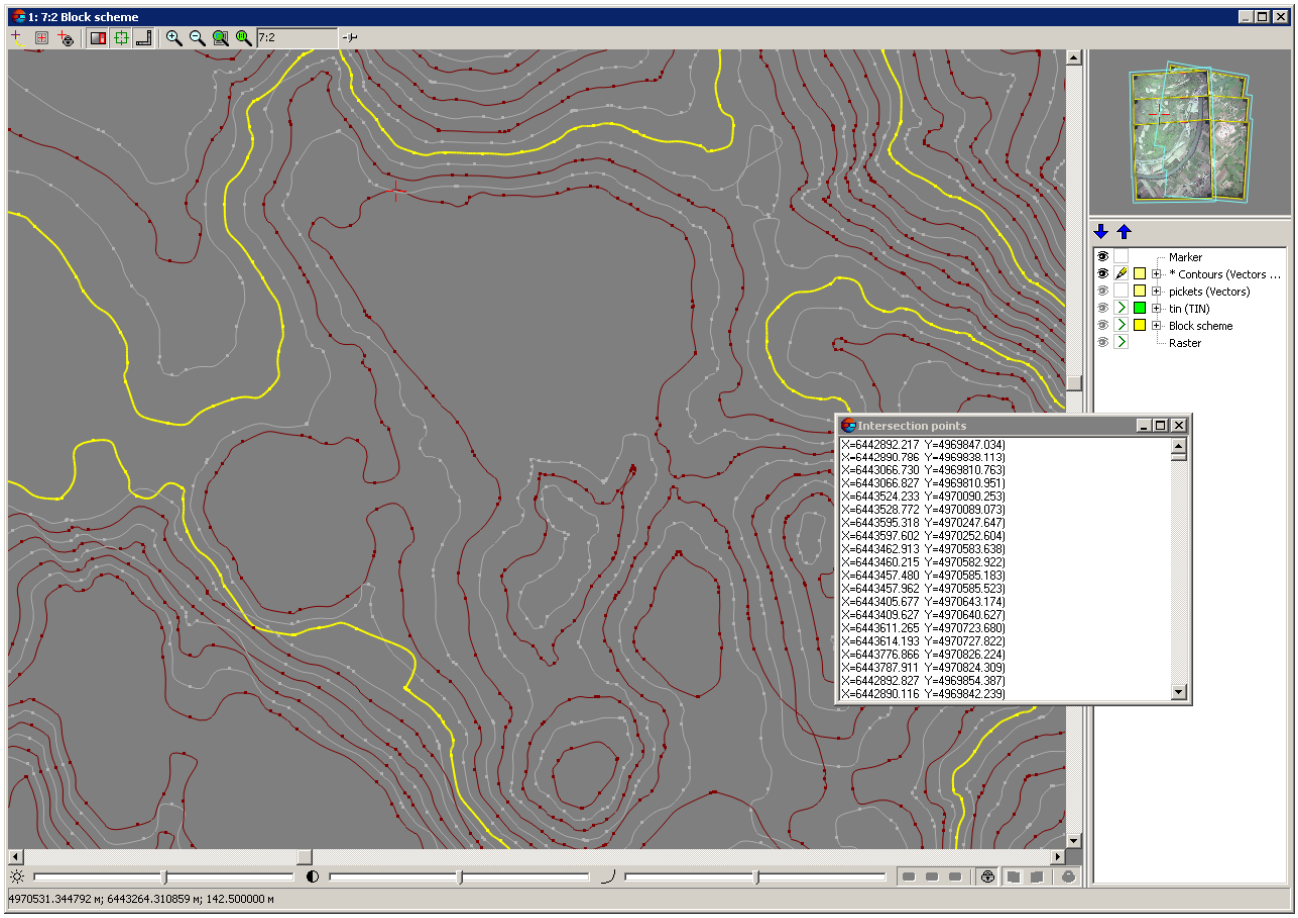
- Control of intersections and self-intersections of contours (see chapter Controlling contours intersections)
- Checking contours against points of loaded vector layer (see chapter Checking contours by points)
- Automatic merging of contours (see chapter Merging contours)
- Control of contours merging (see chapter Checking contours merging) – searching and eliminating gaps between contours.

7.6.1. Controlling contours intersections

Smoothing of contours may lead to appearance of their intersections and self-intersections.

To check the contours for intersections, execute the menu command **Vectors | Contour operations | Search for contours intersections**. If it is necessary to check only neighboring contours, check the **Neighbor contours only** option. When the search process is complete, the list of coordinated of found intersections is opened.

Selecting the point in the list makes marker jump to the point, for consequent editing of the intersection (see chapter Editing of vector objects).

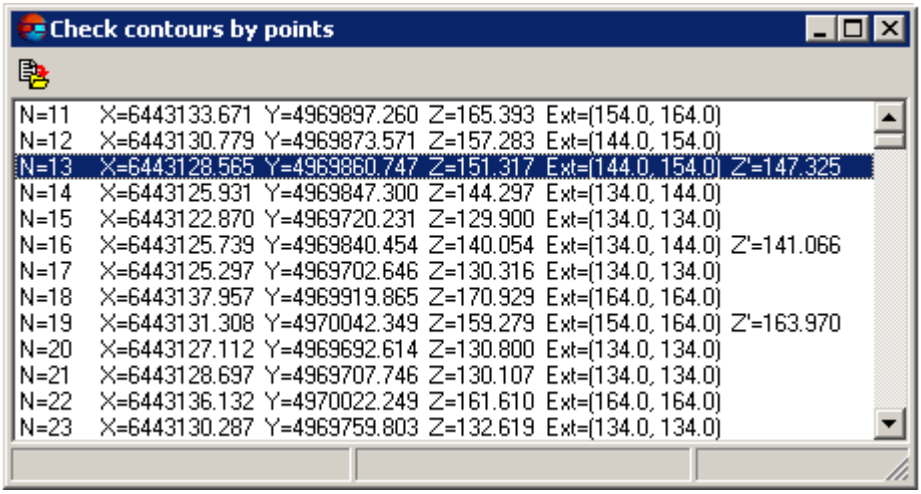


Controlling contours intersections

7.6.2. Checking contours by points

After the contours have been built, they can be verified by comparing with existing pickets for the same area. To perform the verification, open contours and pickets in the different layers and execute the menu command **Vectors | Contour operations | Check contours by points**. In consecutive windows, select the contours layer, points layer and enter maximum Z deviation.
The checking operation is based on the following principle. An internal temporary TIN is build over contour nodes, and each point is compared to the Z coordinate of TIN surface in the X,Y coordinates of the point.
After the process is complete, a list of found error is shown. The list contains the ordinal number **N** and **X, Y, Z** coordinates of the point, Z range (**Ext**) of the neighbor contours, between which the point is found. The last field contains errors found as the result of the verification. The errors belong to one of the two types:

- The point "overflies" (it's Z falls out of the Z range of neighbor contours). In this case the errors field is left empty.
- The difference between the internal TIN and the point is larger than the given maximum value. In this case, the error field shows the computed Z-coordinate of the TIN (**Z'**).



Errors found with maximum deviation set to 1 m

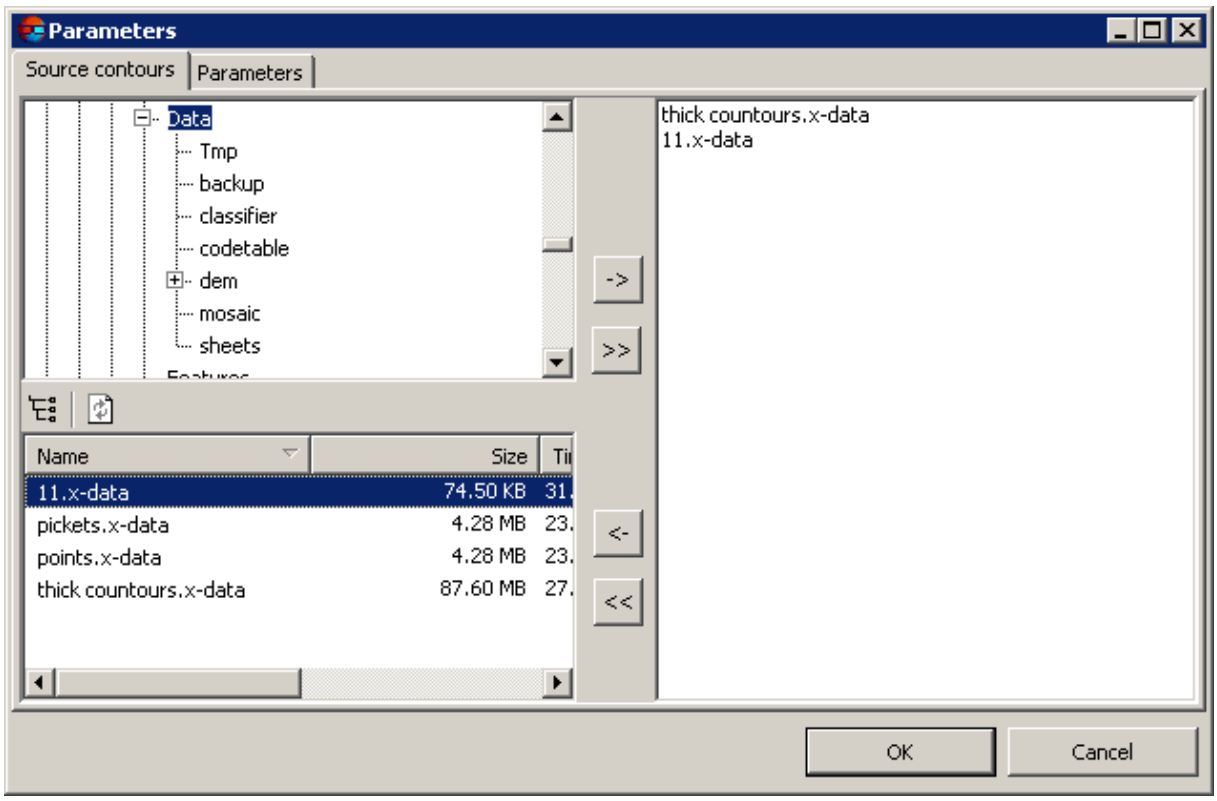
Selecting a row in the list moves marker to the point and centers 2D window on it to make possible editing of contours in stereo mode. See chapter Polyline/polygon editing.

Use the button to save the list to a text file.

7.6.3. Merging contours

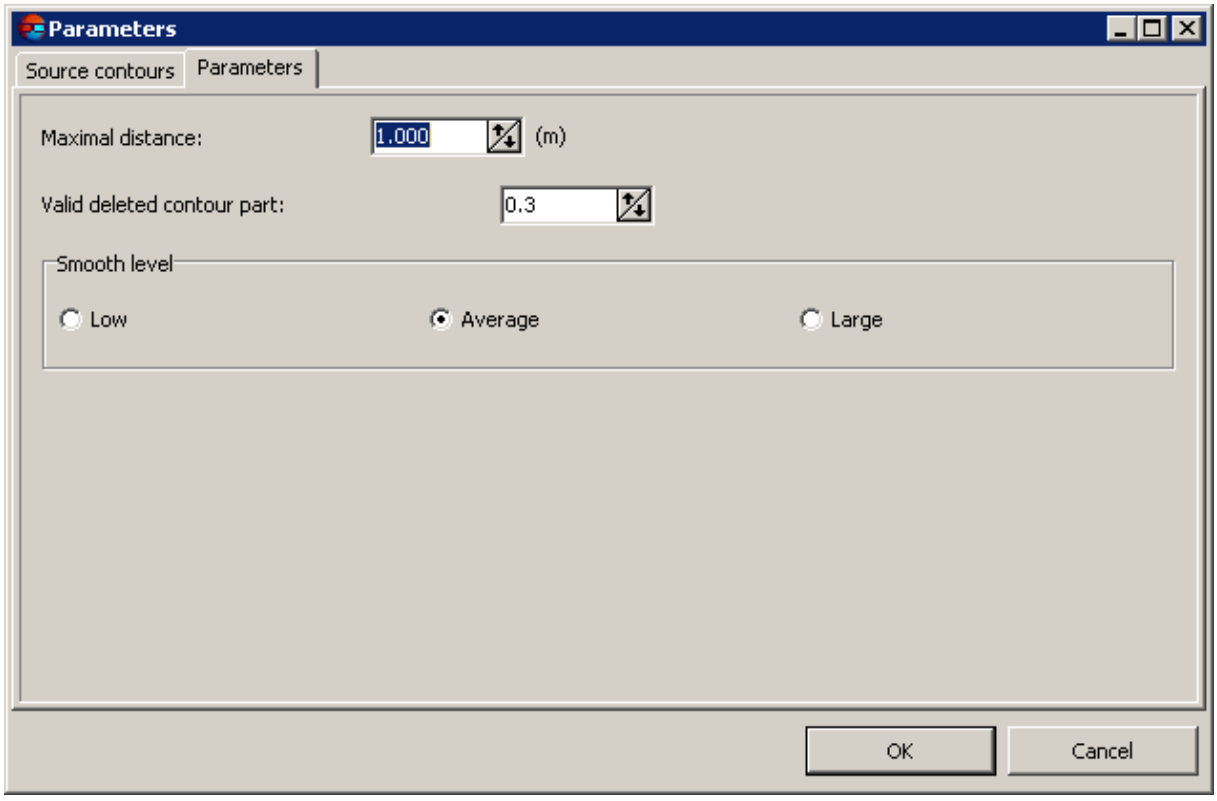
To merge contours manually, open a vector file with additional contours (e.g. from the next stereopair), adding it to the current layer with contours: check the **Add new data to layer** option when opening the resource.
Use vector objects editing tools to perform stitching, in particular snapping (see chapters Polyline continuing and Type of snapping), and Merging polylines tool.

Contour fragments created for different stereopairs of the block may be stitched together automatically. The command **Vectors | Contour operations | Merge contours** is used for doing that. It opens the corresponding **Parameters** window on the **Source contours** tab to select resources with the contours for merging.



Select contour fragments for merging

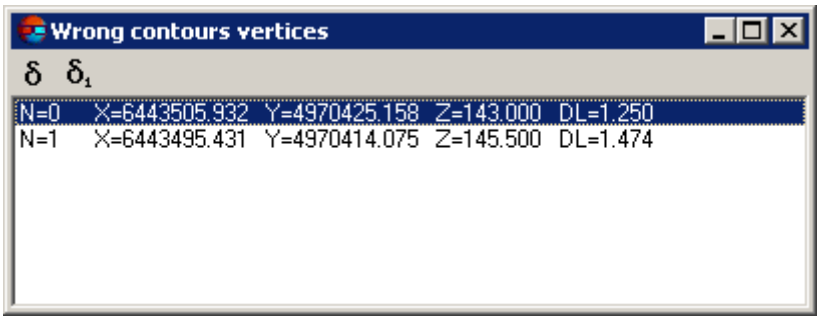
In the bottom left list of the window, select the source vector resources containing contour fragments created from TIN for adjacent stereopairs, and with mouse doubleclick (or the -> button) add the to the right window pane.
On the **Parameters** tab, define the maximum size of gap between contours which should be stitched; allowed deleted contour part (for overshoot intersections) and smoothing level. Intersecting contours will be stitched at the intersection point deleting the ends.



Press **OK**. A new layer with merged contours is created in the Layer Manager.

7.6.4. Checking contours merging

Quality of automatic or manual contours merging may be verified with the operation of checking the merging (**Vectors | Contour operation | Check of contours merging** menu command). The parameter is the **check distance** – maximum size of the distance between contours which is considered the gap. A list of coordinates of found gaps is shown (coordinates of the gap center and its size **DL** for each gap). Selecting a row in the list moves marker to the gap center point for editing the contours.



List of contour gaps after merging

When a gap is selected, it can be fixed by pressing the (**Merge**) button. All gaps can be fixed by pressing the (**Merge all**) button.

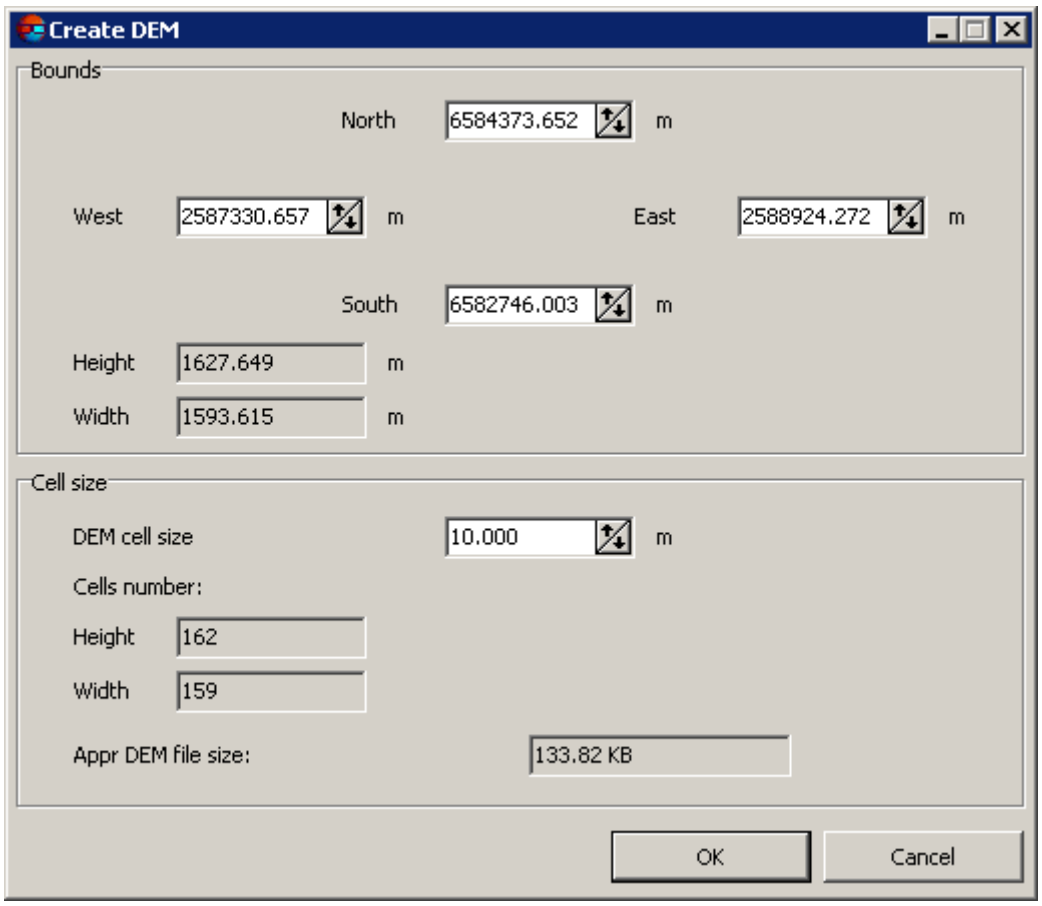
8. Digital Elevation Model (DEM)

8.1. *DEM building*

Initial data for regular grid – **DEM** (Digital Elevation Model) – creation are relief models, represented as irregular network of triangles (TIN) or points as well as any vector objects. See the chapter Building TIN and Pickets creation. All ways of DEM building are described below.

8.1.1. Building DEM using TIN

When operation of DEM creation from TIN (using menu command **DEM | Build | From TIN**), is started, the **Create DEM** dialog for parameters selection is opened.

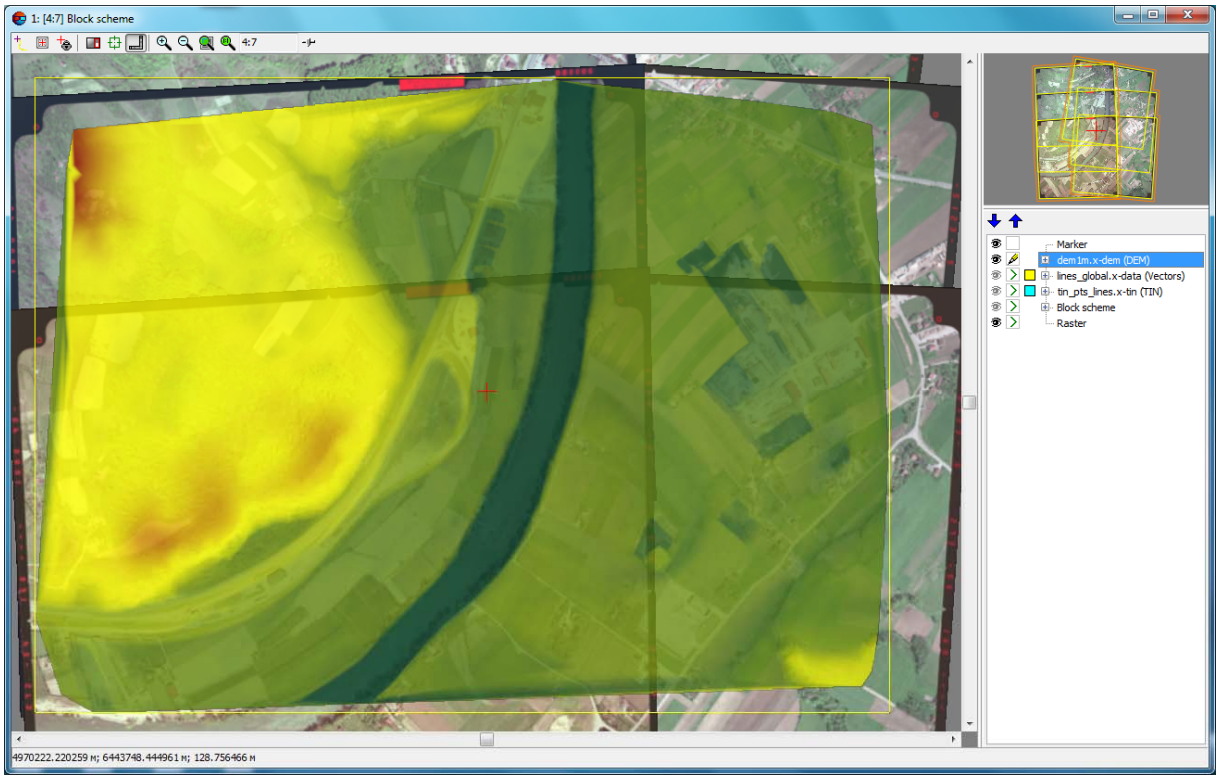


DEM creation parameters from TIN

You can change the bounds coordinates of output DEM in **North**, **West**, **South**, **East** text fields. To do that input new coordinate values there. The defaults are the coordinates of the corners of the rectangle, which describes an area of the source TIN. In the **Height** and **Width** fields you can see calculated sizes of DEM border in meters.

Use the **DEM cell size** parameter to set a size of DEM cell in meters. This value should be commensurable with the size of grid cell used for TIN creation or with the size of the smallest TIN triangle. Too small DEM cell size just causes increasing of the output file size and time of the operation without improving the accuracy. Number of cells for width and height and approximate size of DEM file is calculated using the cell size specified here.

Click **OK**. Specify DEM file name, define target folder in profile resources and click **Save**. Select one of loading options and click **OK** (see also the chapter DEM saving and DEM loading). The operation of DEM building is started. In **Manager** (see the chapter Layer manager) the appropriate layer is created.



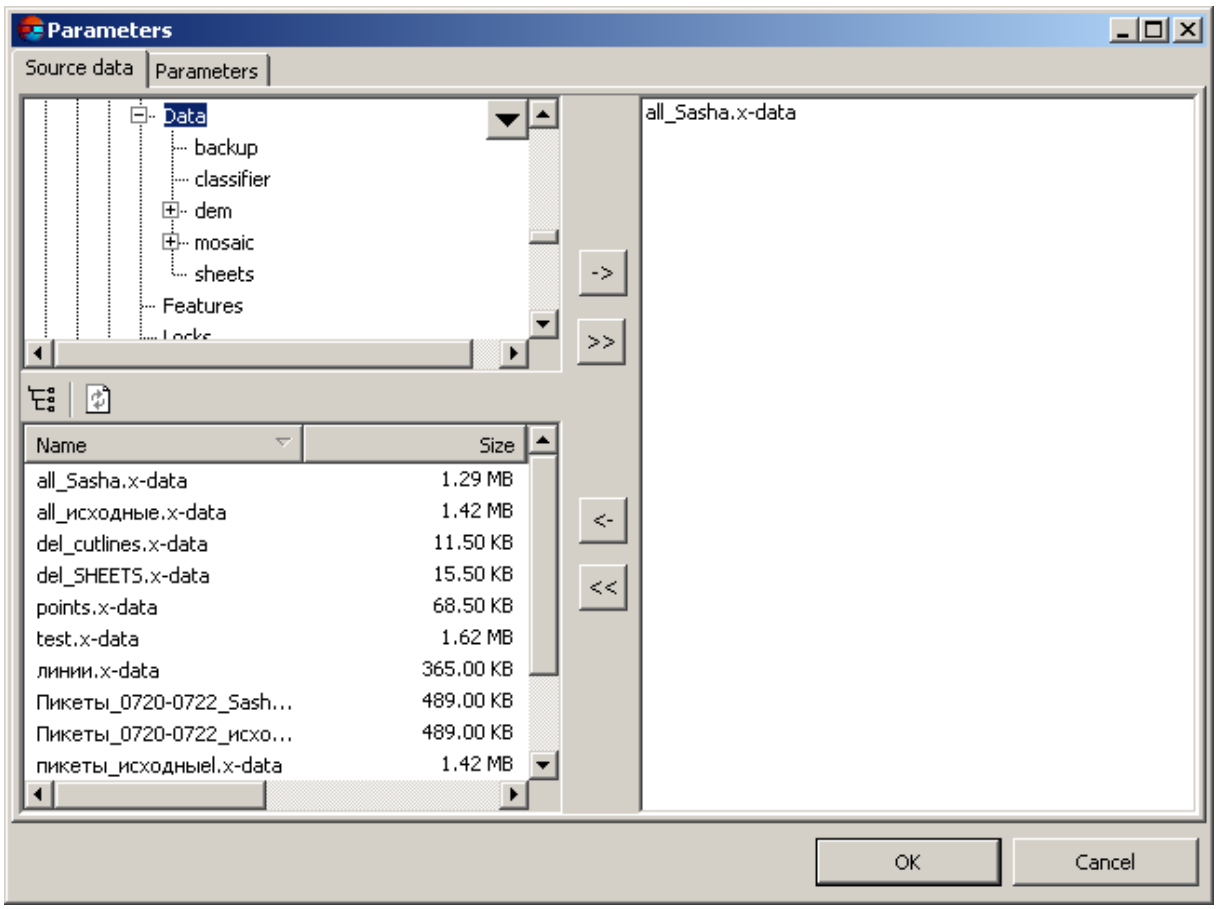
Digital Elevation Model for the whole block, created from TIN

During moving marker over DEM you can see DEM points coordinates and heights in Status bar.

Use the **DEM | Rebuild by TIN** menu command to refresh DEM in case of changing the initial TIN.

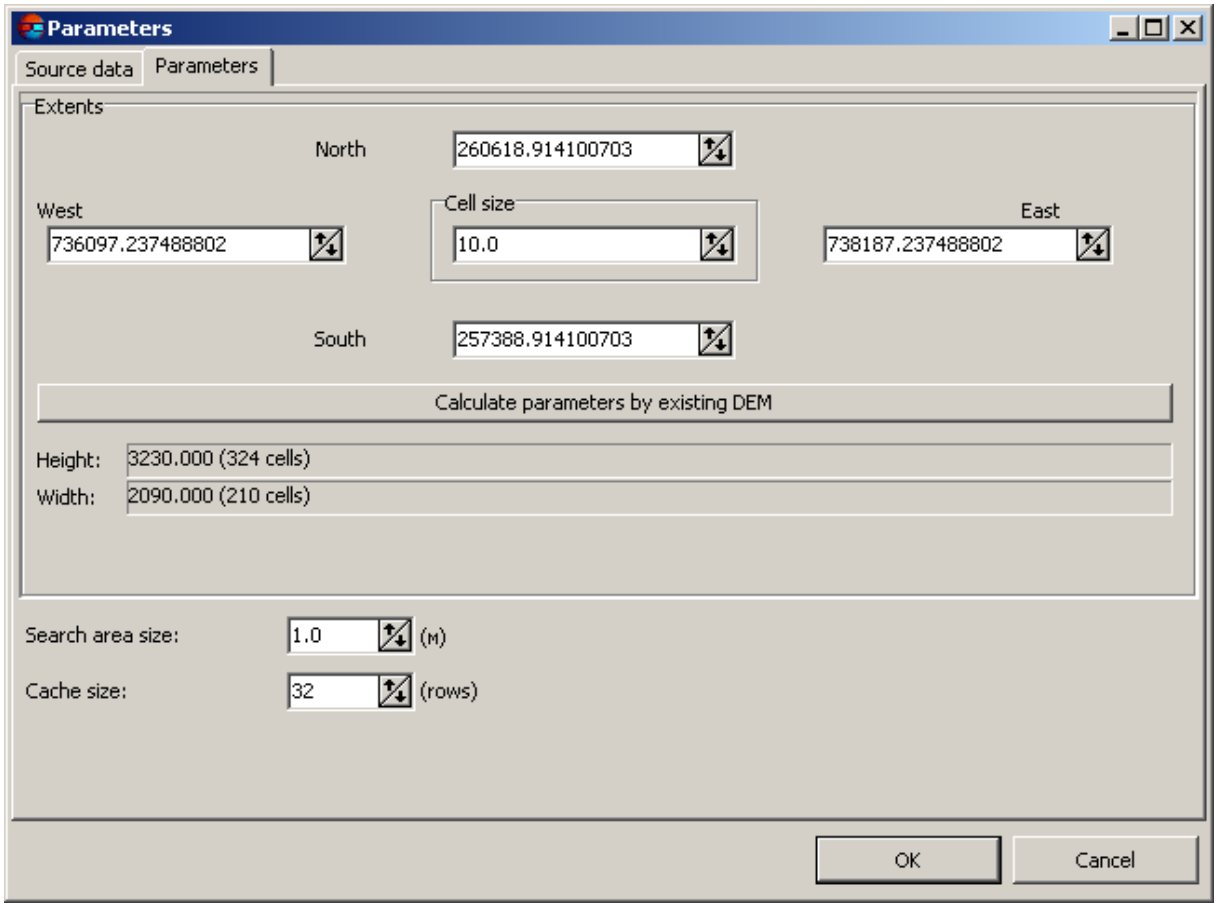
8.1.2. Building DEM using pickets

To start DEM building using available pickets (point objects with XYZ-coordinates) use menu command **DEM | Build | From pickets**. The **Parameters** window is opened, where you can select source data from profile resources – files with pickets, and setup parameters of output DEM.
See also the chapter Pickets creation.



Selection of source pickets

Proceed to the **Parameters** tab and adjust parameters of algorithm of DEM building.



Parameters of output DEM

Algorithm of DEM building from pickets uses method of “smoothing window”, which moves from node to node of DEM over set of pickets and considers heights and weights of the pickets fallen in it. The height of DEM node located in the center of the “window” is defined as total of picket’s heights in the “window” according to their weight.

You can change the bounds coordinates of output DEM in **North**, **West**, **South**, **East** text fields. To do that input new coordinate values there. The defaults are the coordinates of the corners of the rectangle, which describes an area of the source TIN. In the **Height** and **Width** fields you can see calculated sizes of DEM border in meters.

Use the **Cell size** parameter to set a size of DEM cell in meters. This value should be commensurable with the average distance between pickets. Too small DEM cell size just causes increasing of the output file size and time of the operation without improving the accuracy.

The **Calculate parameters by existing DEM** button allows to calculate adjacent boundary using neighboring DEM (if any). After pushing the button the **Open** window appears, where you should select the neighboring DEM.

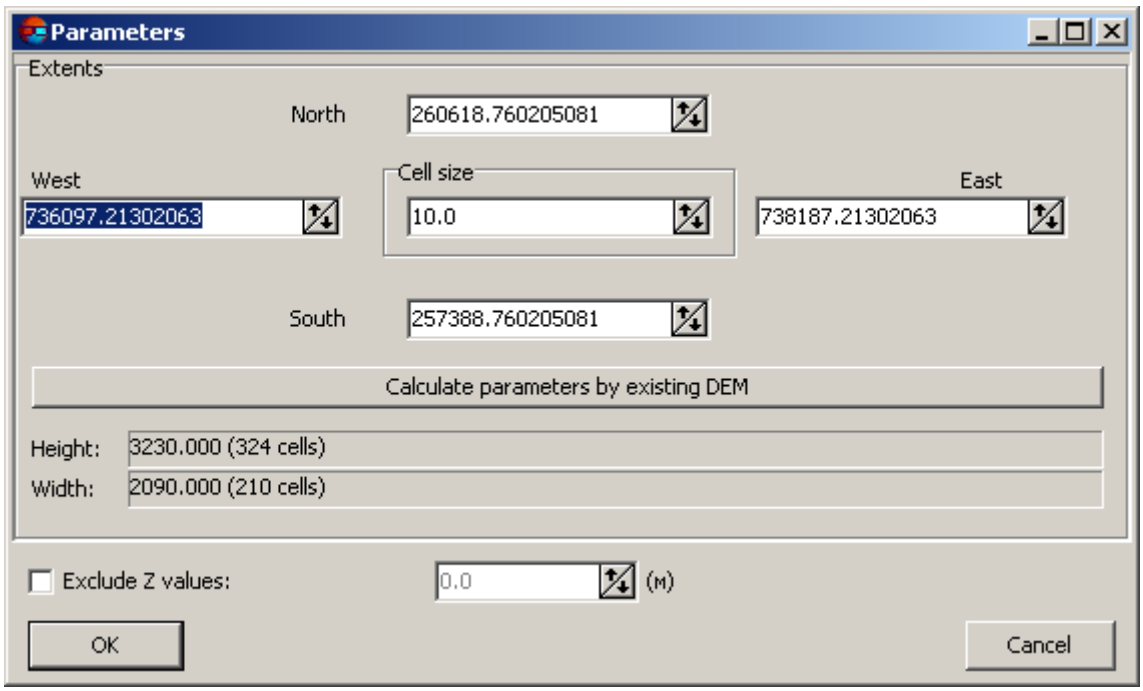
The **Search area size** parameter defines radius of the “smoothing window” (in meters on terrain). Make sure that the **cell size** of DEM to be created does not exceed a diameter of the “smoothing window”.

Click **OK**. Specify DEM file name, define target folder in profile resources and click **Save**. Select one of loading options and click **OK** (see also the chapter DEM saving and DEM loading). The operation of DEM building is started. In **Manager** (see the chapter Layer manager) the appropriate layer is created

8.1.3. Building DEM using regular pickets

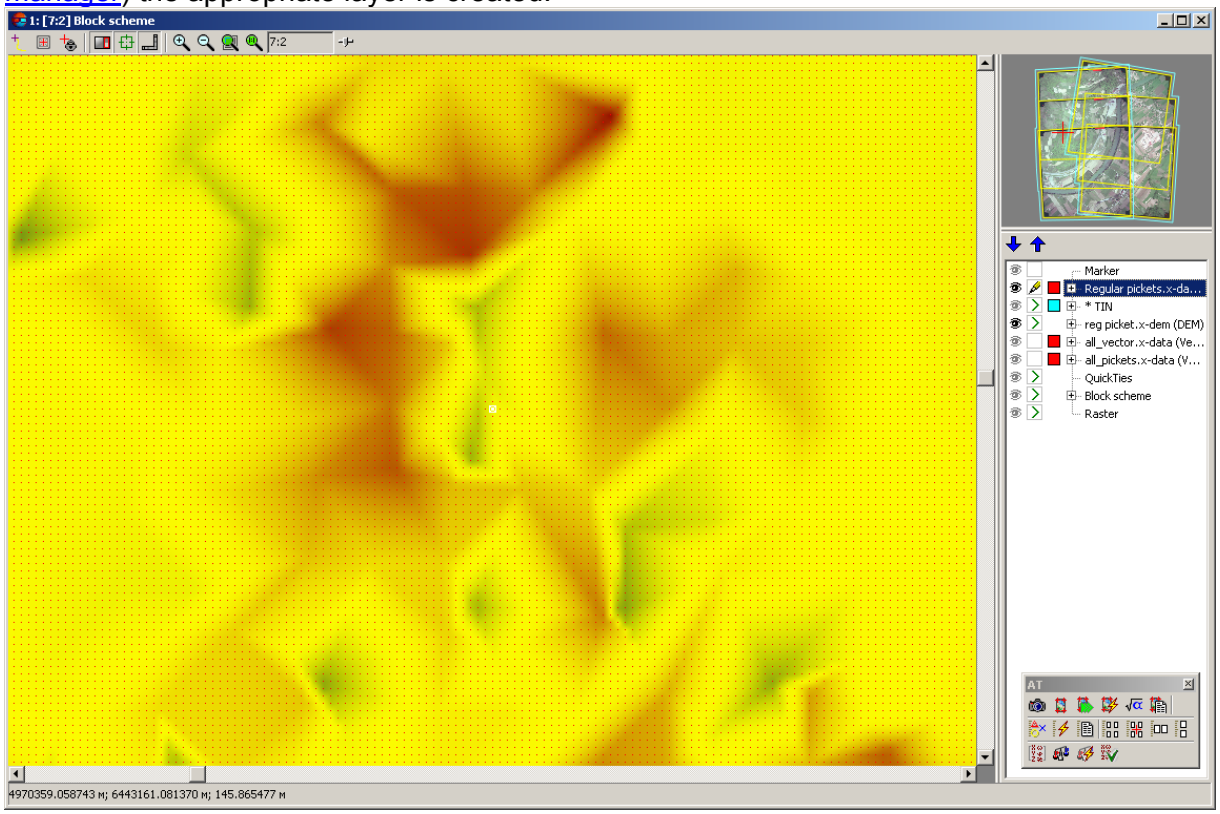
If there is a set of regular pickets created in nodes of regular grid (see the chapter Creation of pickets in pathway mode) or imported from external files, you can use them for DEM building. Load pickets to separate layer, make it active. Select menu command **DEM | Build | From regular pickets**. The **Parameters** window is opened. The settings in the **Extents** panel are the same as parameters of DEM building from pickets (see the chapter Building DEM using pickets). In addition, you can exclude from the operation pickets with specified height (by enabling the option **Exclude Z values**).

© 2011



Parameters of DEM creation from regular pickets

Click **OK**. Specify DEM file name, define target folder in profile resources and click **Save**. Select one of loading options and click **OK** (see also the chapter DEM saving and DEM loading). The operation of DEM building is started. In **Manager** (see the chapter Layer manager) the appropriate layer is created.



DEM created from regular pickets

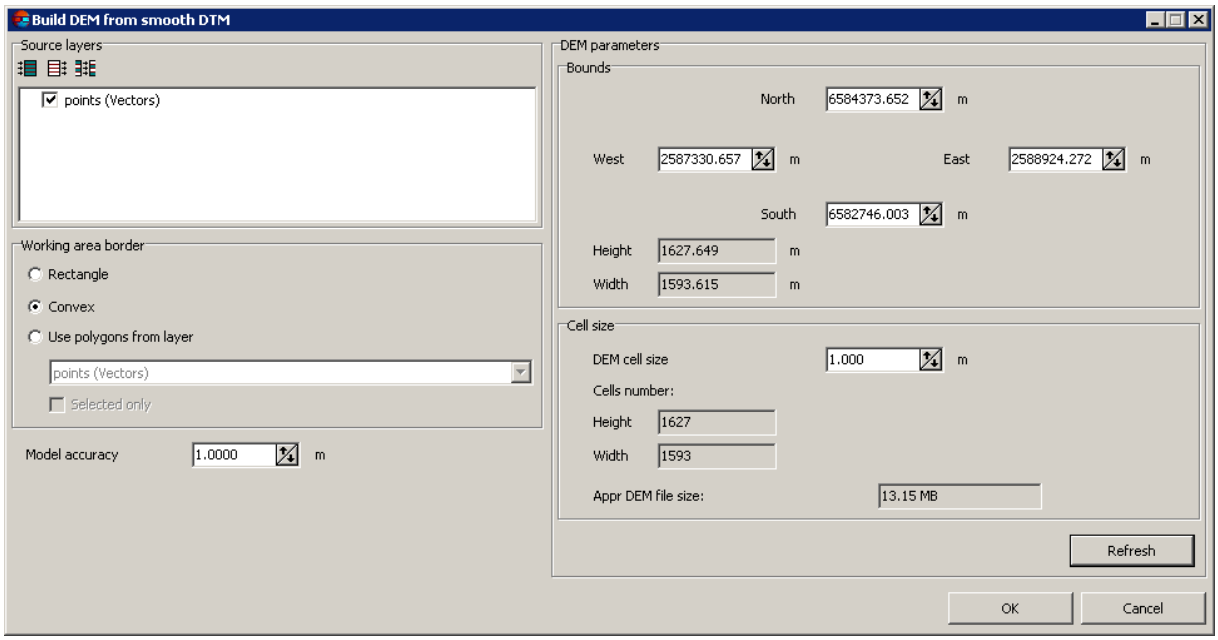
8.1.4. Creating smooth model DEM

A DEM can be created as smooth model from objects of selected vector layers. The smooth model implies creation of DTM with the given precision.

To create DEM, open vector layers and execute menu command **DEM | Build | Smooth model**. The **Build DEM from smooth DTM** dialog appears.
In **Source layers** section select base vector layers by checking them in the list.
The **Working area border** allows settings the DEM border:

- **Rectangle**
- **Convex**
- **Use polygons from layer** – use all or selected polygons from the chosen layers

The **Model accuracy** setting defines maximum Z deviation of DEM from points and vertices of vector objects, judging from the accuracy of vectorization in stereo mode with the help of correlator. As a rule, the precision is about 1 pixel. Pixel size (GSD) can be seen e.g. in the grid properties dialog (**Grid | Properties**).



The **DEM parameters** section allows setting the output DEM limits and size. The **Cell size** section is used to set the DEM cell size and assess the number of cells and DEM file size.

8.1.5. Creation of DTM by dense model

The system provides for the possibility of building a DTM by dense model, cell size of which corresponds to one pixel of the image.

> **Warning: The procedure of building a DTM by dense model is performed only for adjusted stereo pairs. I.e it is necessary to adjust the stereo pairs in the free or georeferenced model previously (see Aerial triangulation User Manual). Otherwise, the DTM will not be generated or will be generated incorrectly.**

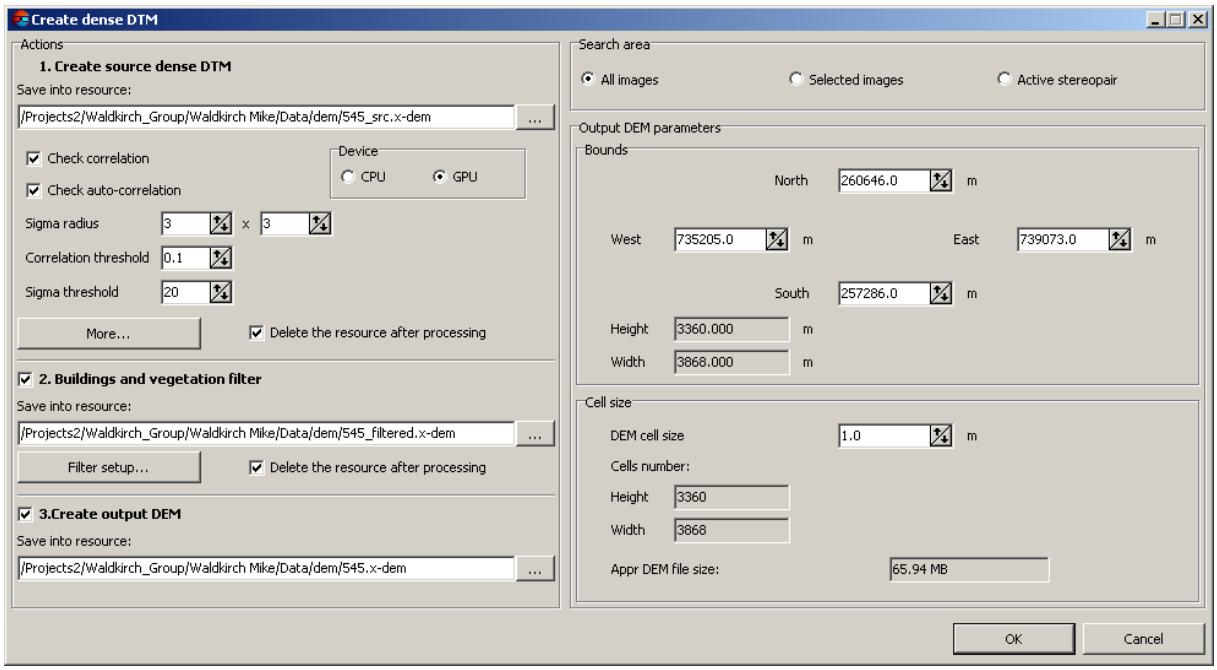
Follow these steps to build the DTM by dense model:

1. Specify the DEM building area using the grid. Since the calculation of the coordinates will be implemented in each pixel of the selected area, then the grid nodes are not taken into account, grid only specifies the boundary of the DEM. Select the **Grid | Create** menu command to create a grid (see chapter Grid creation).

2. Perform the **DEM | Build | Dense DTM** menu command. If the grid was not built previously, then on performing the command the grid is created on the entire block of

images and **Grid properties** window will be opened. Push the **OK** button if you intend to build the DEM at all the stereopairs of block. Push the **Cancel** button to create the grid with other boundaries, using **Grid | Create** and repeat the **DEM | Build | Dense DTM** menu command. Next, the **Save** window opens.

3. Define the folder of active profile and enter the file name of output DEM. Push the **Save** button. **Create dense DTM** window opens.



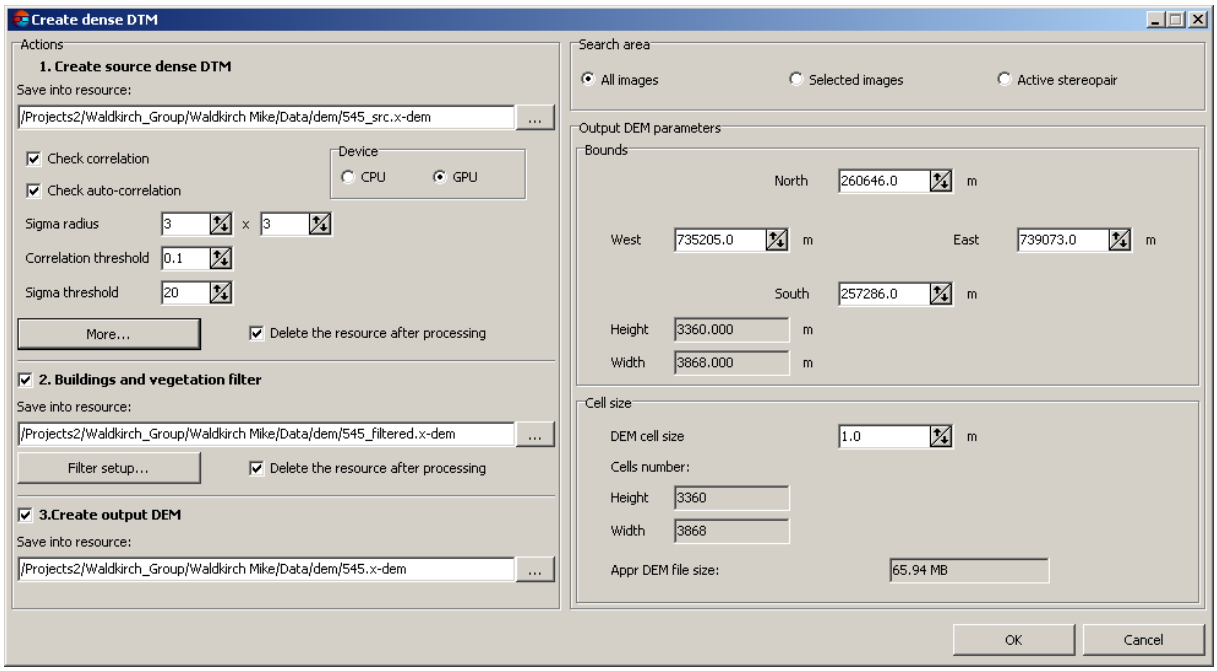
4. Specify the building area (if necessary). Set the parameters of DTM by dense model, if necessary, set the filtering options, as well as boundaries and / or cell size for the output DEM. See the detailed description in chapter Building parameters of DTM by dense model. Push the **OK** button. Process of DEM building starts, which depending on the size of the building area can take a long time. In the presence of **NVIDIA CUDA** processor provides the possibility of using the **GPU** to accelerate the building of DEMs. See detailed information on the site http://www.nvidia.ru/object/cuda_gpus_ru.html.

8.1.5.1. *Building parameters of DTM by dense model*

Create dense DTM window contains two main parts. The **Actions** panel is located on the left side, which allows to perform the following steps:

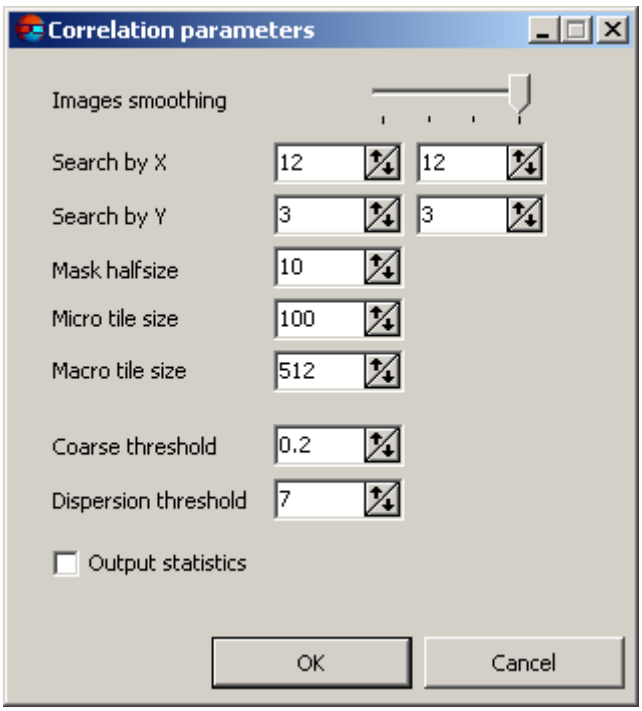
- Building of DTM by dense model;
- Applying the **Buildings and vegetations filter** to built of DTM by dense model (optional);
- Creation output DEM with the specified cell size and / or with the DEM boundaries (optional).

The right panel allows to specify the building area of DTM by dense model, and set also the parameters of output DEM.



To create of DTM by dense model, follow these steps:

1. Specify the building area in the **Search area** panel: **All images**, **Selected images**, or **Active stereo pair**.
2. To save DTM by dense model into the active profile, specify the file name and its location in the **Save into resource** entry line and uncheck the **Delete the resource after processing**.
3. Specify the hardware resource, which to be used for calculations: the central processor (**CPU**) or NVIDIA CUDA processor (**GPU**). Usage the NVIDIA CUDA processor (if any) is significantly reducing the processing time.
4. Switch on the **Check correlation** and **Check auto-correlation** options to verify the correlation on specifying limits in parameters and autocorrelation, respectively.
5. Specify the **Sigma radius** and **Sigma threshold** parameters. Filter on the radial autocorrelation rejects the points, which has the **σ** parameter greater than specified radius. Thus, in particular, the points on the vector objects are "rejected». See also the description of the autocorrelation parameters in chapter Settings of correlator preset.
6. Define the **Correlation threshold** - the minimum allowed value of the correlation coefficient for the matched image pixels. The **Correlation threshold** is set to 0,1 by default.
7. Push the **More** button to define the additional parameters. **Correlation parameters** window opens.

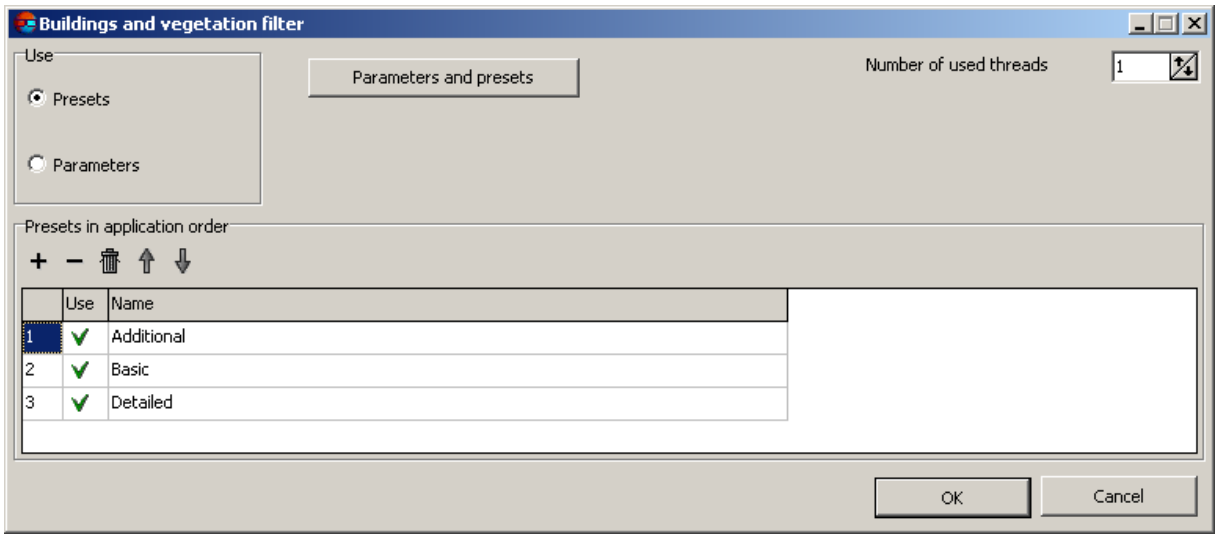


Additional correlation parameters window

Parameter	Description
Images smoothing	The slider serves to define the smoothing degree of images.
Search by X	The size of search area along X axis with respect to initial approximation, calculated on the basis of orientation data.
Search by Y	The size of search area along Y axis with respect to initial approximation, calculated on the basis of orientation data.
Mask halfsize	Half of correlation mask linear size in pixels along X and Y axes.
Micro tile size	Micro tile size - the size of area in macro-tile to match the pixels.
Macro tile size	Macro tile size – the size of image area.
Coarse threshold	The value of correlation coefficient when matching the micro-tiles.
Dispersion threshold	The value of contrast during the matching of the images pixels.
Output statistics	Check box to display the information about the building of DTM by dense model in progress window.

Ability the processing of created DTM by dense model using the Buildings and vegetation filter is provided in the **Create dense DTM** window. To filter the created DEM, follow these steps:

1. Check the **Buildings and vegetations filter** option.
2. *To save* the processed DTM by dense model, specify the file name and its location into the active profile and *uncheck* the **Delete the resource after processing**.
3. Push the **Filter setup** for settings the filtering parameters. **Buildings and vegetation filter** window opens. See chapter Buildings and vegetation filter.
4. Determine the value of **Number of used threads** parameter to distribute processing between the processor cores (If the computer has multi-core CPU or several CPUs).



Filtering settings

Ability the specifying of parameters for output DEM: boundaries and cell size, also is provided in the **Create dense DTM** window.
To save the output DEM, follow these steps:
1. Set the parameters of output DEM in the **Output DEM parameters** panel: specify the boundaries and / or cell size of the DEM.
2. Check the **Create output DEM** option and specify the file name and its location into the active profile to save the output DEM.

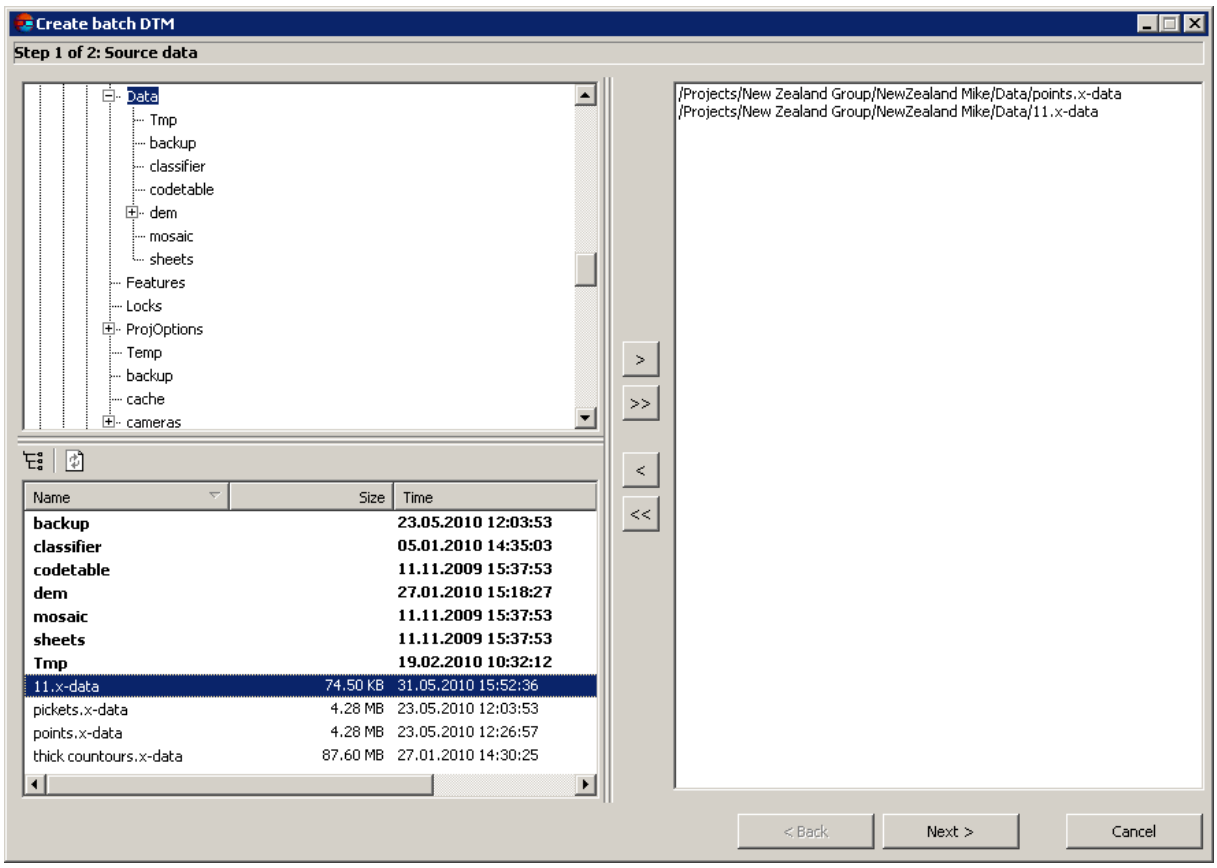
Push the **OK** button in the **Create dense DTM** window after the defining all parameters. The process of building the DTM by dense model starts.

8.1.6. Batch DTM creation

Batch creation allows creating DTM (in TIN or DEM form) in case the volume of source data (millions of points/vertices) surpasses the capabilities of the hardware for immediate DTM creations.

The batch DTM creation approach is as follows: the source vector resources are "cut" into sheets, each of the sheets is used to build TIN and DEM, which in turn are merged into output DEM.

Execute the **DEM | Batch DTM creation** command. The **Create batch DTM** window appears, open on the tab **Step 1: Source data**. Select vector resources. Using buttons with arrows, create the list of all data form DTM creation. Press **Next**.



Creating source vector data list for batch DTM creation

The window switches to the **Step 2: Options** tab.
The **Processing** section is used to set the tasks which are executed:

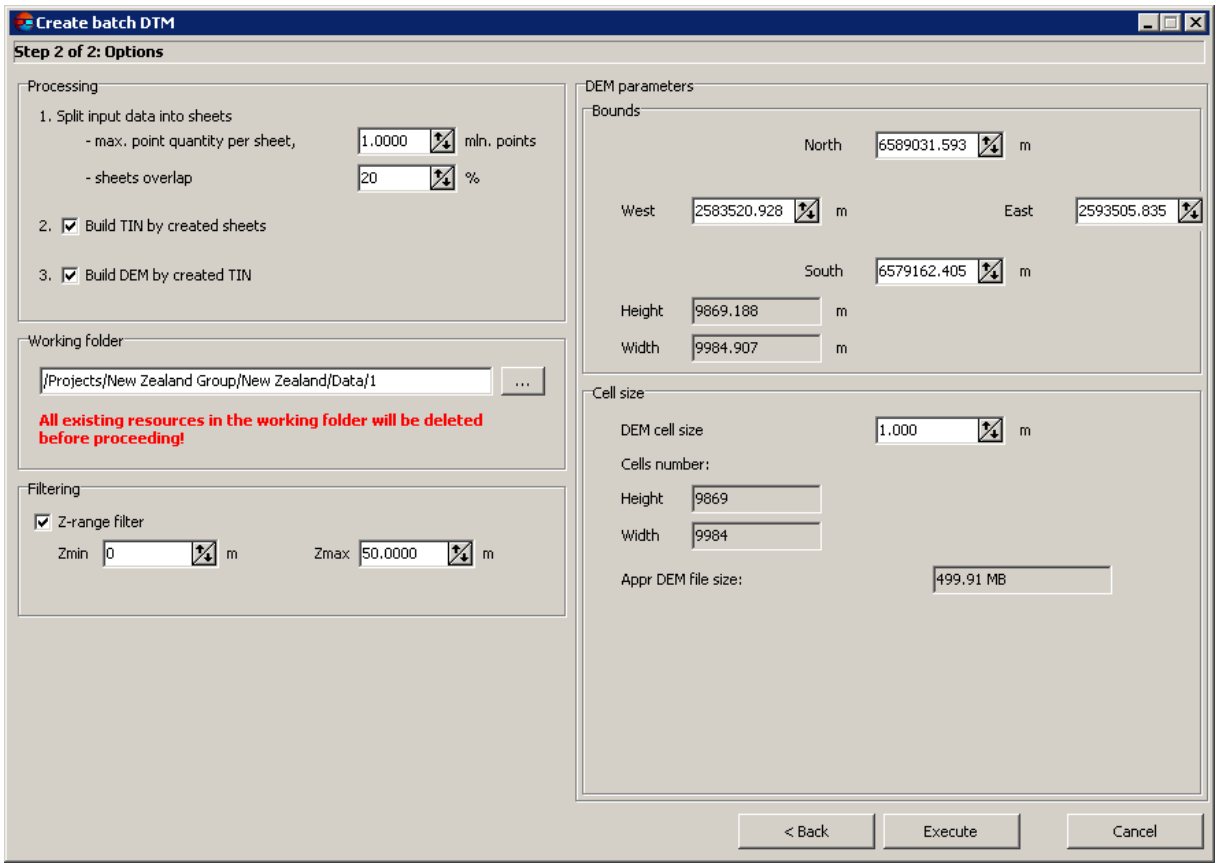
1. Split input data into sheets – define maximum number of points/vertices in each list (1 million points by default) and sheets overlap in %. The number of sheets is deduced from this parameters; in output folder a vector resource is created for each sheet with extension *.x-data* and name *sheet_(sheet number)*.
2. **Build TIN by created sheets** – turn this on to create TIN resources for each sheet. This task leads to creating TIN files with *.x-tin* extension and *sheet_(sheet number)* name in output folder.
3. **Build DEM by created TIN** – turn this option on to build united DEM from all the TINs created from vector sheets. Checking this option also turns on the **Build TIN** option. This task results in the *dem.x-dem* resource created in the output folder.

In the **Working folder** section define an empty folder for placing output data.

The **Filtering** section allows applying filtration of vector points and respective TIN by Z range. This means ignoring in TIN creation of vector points and polyline/polygon vertices, Z coordinate of which falls outside the given range.

The **DEM parameters** section allows changing limits and cell size of the united DEM and assessing its number of cells and approximate file size.

After all the parameters are set, press the **Execute** button. A confirmation message is shown about deleting all existing data in the output working folder. After the process is complete, the output DEM (if the corresponding options is checked) is loaded into a new layer in Layer manager.

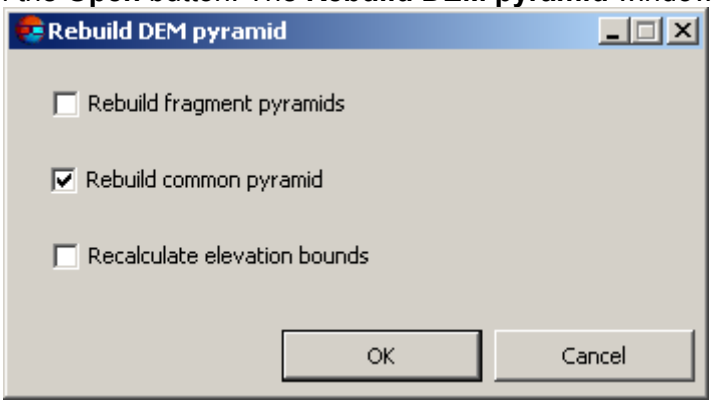


8.2. *DEM rebuilding by TIN*

DEM | Rebuild by TIN menu command allows to refresh DEM in case of changing the initial TIN (see the chapter TIN editing).

8.3. *DEM recovering*

The system provides the ability to recover the DEM integrity, which may be corrupted as a result of various operations or the DEM filtering.
Use the **DEM | Recover menu** command to recover the DEM. The **Open** window opens. Determine the name and location of the recovered output DEM into the active profile resources and push the **Open** button. The **Rebuild DEM pyramid** window opens.



Window consists of following parameters:

- **Rebuild fragment pyramids** – allows to rebuild the DEM by fragments, in case the DEM consists of these;
- **Rebuild common pyramid** – allows to rebuild the common DEM pyramid;

- **Recalculate elevation bounds** – allows to “repaint” the DEM after the changing of its bounds by Z-range.

8.4. *DEM saving*

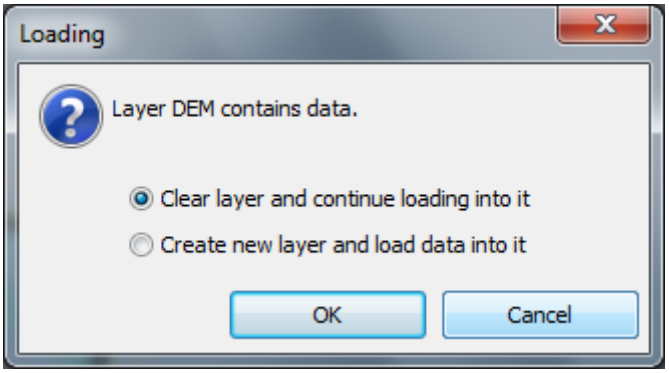
DEM is saved to file with *x-dem* extension. File name and target folder in profile resources is defined prior to DEM creation, so the DEM is saved automatically to file specified just after creation is complete.
The **DEM** menu item contains the following commands:

- **Save copy** – allows creating a copy of active DEM – i.e. to save current DEM to specified file in profile resources.
- **Save selection** – allows to save selected (using the **Rectangle selection mode** tool) DEM area to specified file in profile resources.
- **Export** – allows exporting created DEM to external files of different formats.

8.5. *DEM loading*

For DEM loading you can use the following commands of **DEM** menu item:

- **Load** – allows to open DEM saved in profile resources (file with *x-dem* extension). After that the **Loading** window is opened where you can select one of the loading options, if there is DEM layer in **Manager**:
  – **Clear layer and continue loading into it** – loads DEM to active (or last opened) layer with DEM, after clearing the layer from existing data.
  – **Create new layer and load data into it** – creates new layer with file name for DEM loading.



Options of DEM loading

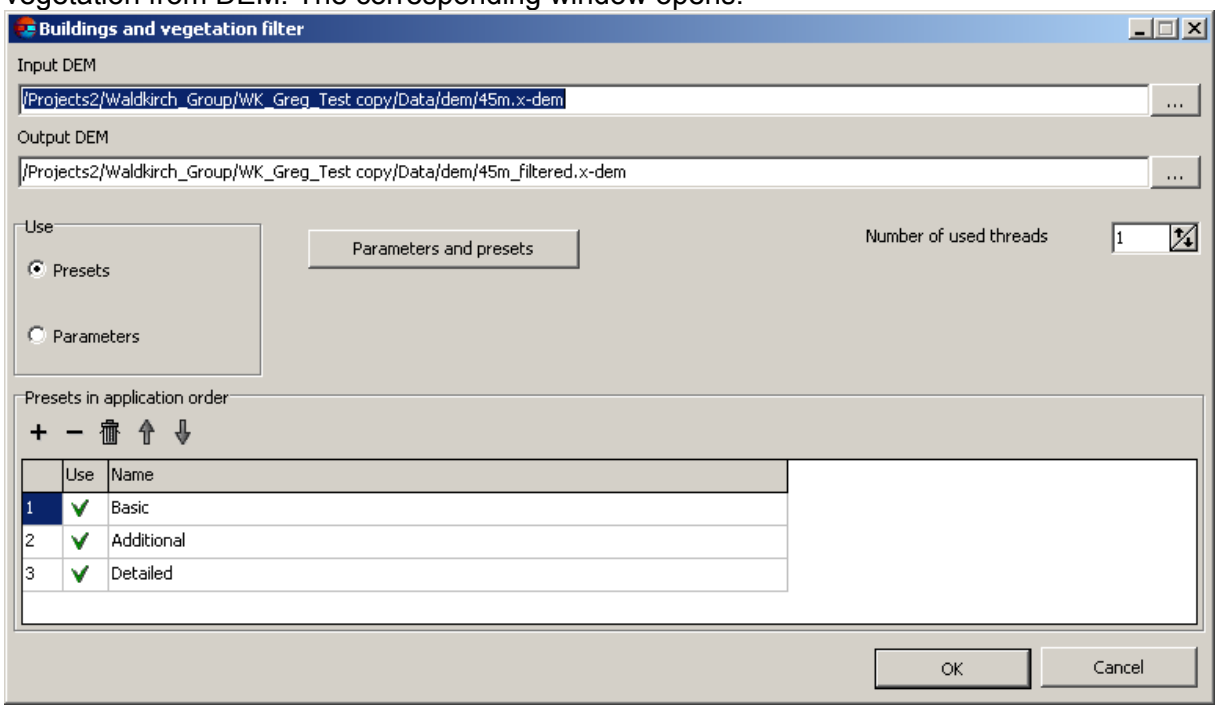
- **Open from file** – allows to open DEM directly from external source, without conversion to internal format (*x-dem*). At that, if there is DEM layer in **Manager**, you should select one of the following loading options – **Clear layer and continue loading into it** and **Clear layer and continue loading into it**. You can save opened DEM to internal format *x-dem* in profile resources.
- **Import** – allows to import DEM from external files of different formats. See the chapter Import of DEM.
- **Load LIDAR data** – allows to transform the LIDAR data to DEM. See chapter LIDAR data loading.

8.6. *DEM filtration*

8.6.1. Buildings and vegetation filter

Buildings and vegetation filter is provided for DEM. The operation principle of the filter is as follows. Base points with a given decimation ratio are computed for DEM. Filter is applied to this set of points to remove the points, which fall onto the houses, trees, cars etc. At that, random peaks are also filtered. As a result, there are only the points, describing the terrain. Next, editing of the DEM is performed on the basis of the "relief" points, and the DEM is saved in a new layer.

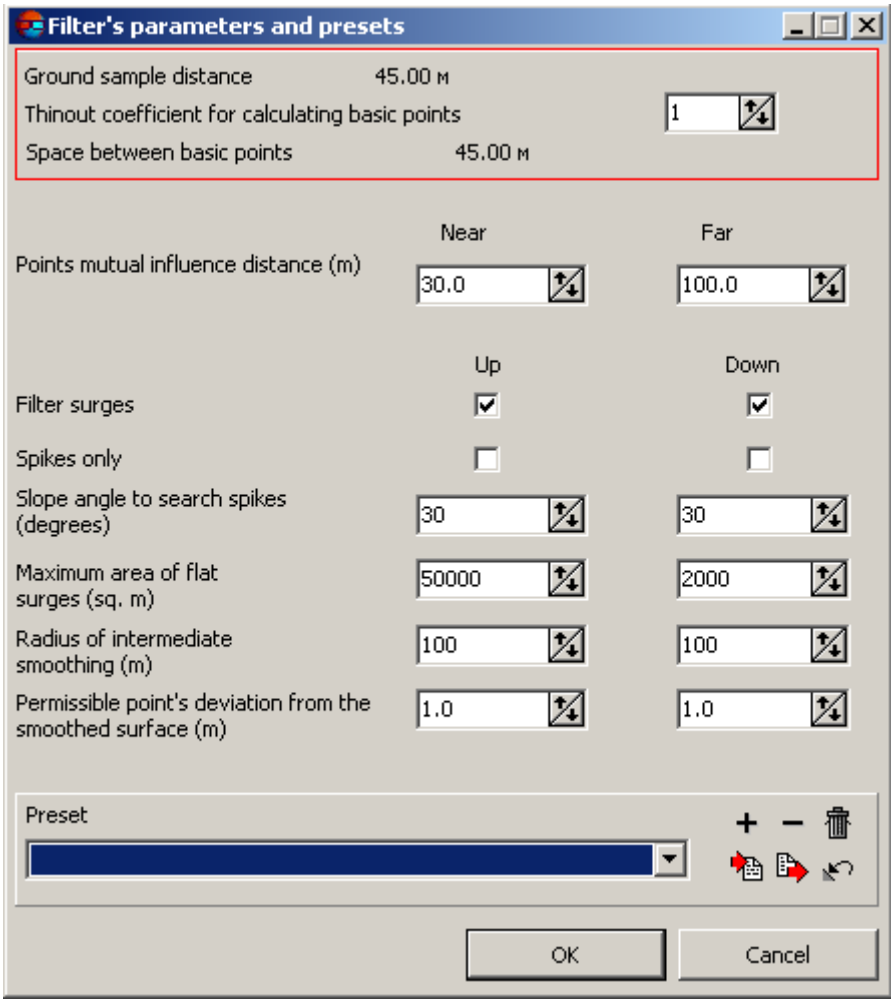
Use the **DEM | Buildings and vegetation filter** menu command to filter the buildings and vegetation from DEM. The corresponding window opens.



Buildings and vegetation filtration from DEM

Since the filter is applied to the points, which computed from DEM, the detailed description of the filter parameters, see chapter Filter of buildings and vegetation. In the case of DEM filtering by default it is proposed the serial passage only the two stages of filtration: the basic and additional filtration.

The thinout coefficient for finding the basic pickets from DEM is given in the **Filter's parameters and presets** window.


Filter's parameters and presets
Ground sample distance 45.00 м
Thinout coefficient for calculating basic points 1
Space between basic points 45.00 м
Near
Far
Points mutual influence distance (m) 30.0 100.0
Up
Down
Filter surges
Spikes only
Slope angle to search spikes (degrees) 30 30
Maximum area of flat surges (sq. m) 50000 2000
Radius of intermediate smoothing (m) 100 100
Permissible point's deviation from the smoothed surface (m) 1.0 1.0
Preset
OK
Cancel

Determination of thinning parameters for finding the basis points

The detailed description of other window options, see chapter Setting up parameters of filter.

8.7. *Operations on DEM*

8.7.1. Merging DEMs

For merging of two adjacent DEM's, which have overlap area, use menu command **DEM | Merge**. After selecting the command the **DEM merging** window is opened that is used for selecting of initial DEM's (files with *x-dem* extension) from profile resources, and also for defining of output DEM parameters (see the chapter Building DEM using pickets). In addition, you can specify actions in overlap areas during merging:

- use data from top DEM;
- use data from bottom DEM;
- calculate and use average values of adjacent DEM's points in overlap area.

8.7.2. Flattening DEM fragment

DEM | Flatten fragment menu command allows to flatten the selected DEM fragment by height. I.e. DEM height mean value of current fragment will be assigned to all cells of selected DEM fragment.

Selection of DEM fragment is performed by polygons. Perform the following to flatten DEM fragment:

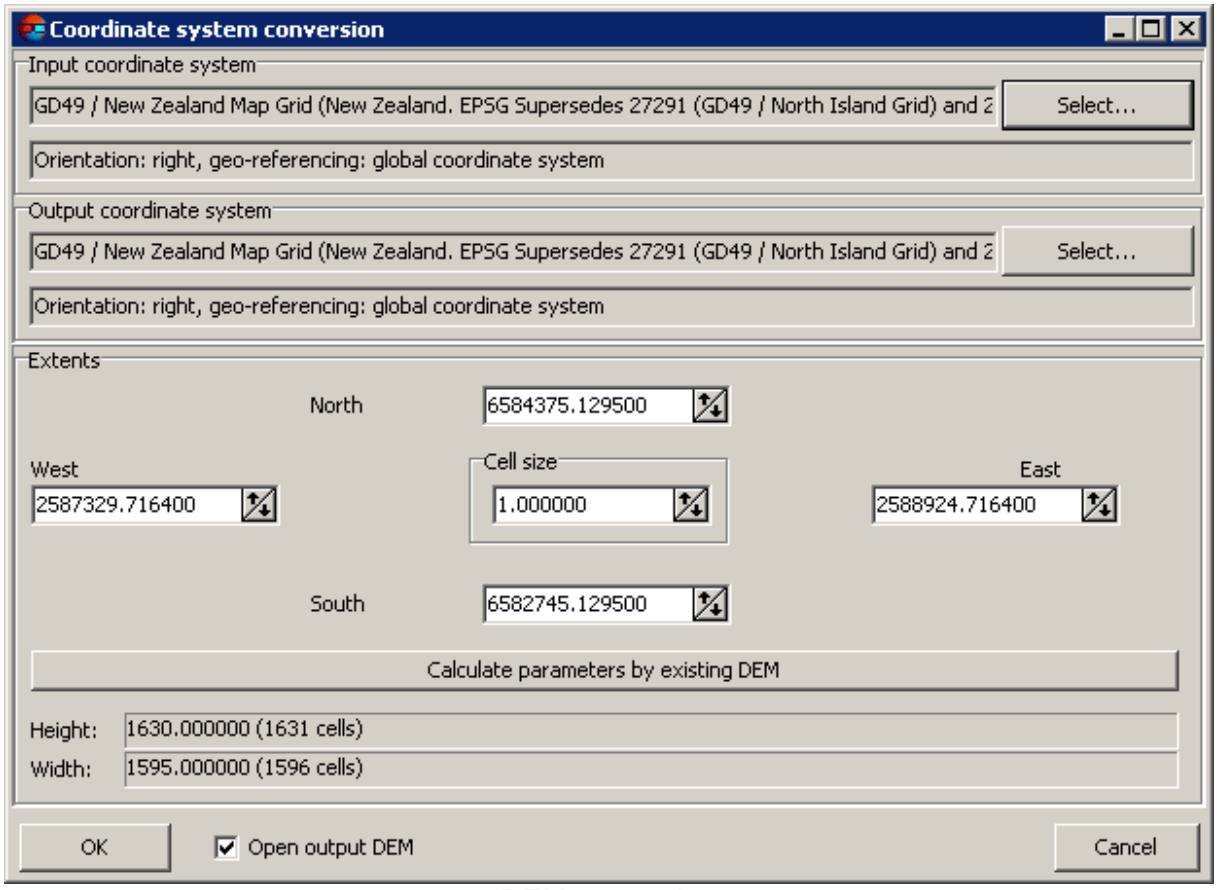
1. Create vector layer;
2. Create polygons;
3. **DEM | Flatten fragment** command.

8.7.3. Transposing DEM

The **DEM | Transpose DEM** menu command allows converting a DEM from left coordinate system into right one and vice versa, i.e. swapping rows and columns, to bring it into compliance with the project. The left/right DEM mismatch may be caused by source vectors, which were used to build the DEM, which should have been XY-swapped (**Vectors | Swap X <–>Y**) before DTM creation but were not.

8.7.4. Converting DEM to another coordinate system

The **DEM | Transform DEM to another CS** allows changing the DEM coordinate system to bring it into compliance with the project coordinate system. Also such need may arise for exporting DEM to another system. The **Coordinate system conversion** is shown on the command execution:



DEM conversion

In the **Input coordinate system** section, specify the actual DEM coordinate system (if it differs from the project coordinate system) with the **Select** button.
Select the output coordinate system in the corresponding section.
If necessary, change limits and cell size of the output DEM, or press **Calculate parameters by existing DEM** button to calculate these parameters as compatible with another existing DEM.
If needed, check the **Open output DEM** option. Press **OK**. Specify the resource for the output DEM.

8.7.5. Transforming DEM by a set of points

There is capability of transforming a DEM by existing set of points by referencing DEM to points coordinates. At that, if points are given by 2 coordinates X and Y, the 2D (plane) transform is used, if they are given by 3 coordinates X,Y,Z – the 3D transform is used.
To perform the transformation, a text file must be prepared containing points coordinates in DEM and corresponding target coordinates. The text file must contain data in either of the following forms:

- lines containing 4 coordinates X Y X* Y* to transform DEM in XY plane;
- lines containing 6 coordinates X Y Z X* Y* Z* to transform DEM in 3D space.

Here, X Y Z are coordinates of a point in DEM, X* Y* Z* are the target coordinates of the same point. The fields can be separated by spaces or commas.

2D transformation implies:

- shift, when only one line with point coordinates is present;
- shift, scale and rotation – when two points are given;
- affine transform – for 3 points;
- projective transform – for 4 points or more.

When 3D coordinates are given, the shift by Z is added to the corresponding 2D transformation (the Z delta value is calculated as mean delta for all the points).

When the text file is ready, activate the DEM layer and execute menu command **DEM | Transform DEM by points set**. Select the text file with points coordinates and press **OK**.

8.7.6. Filtration

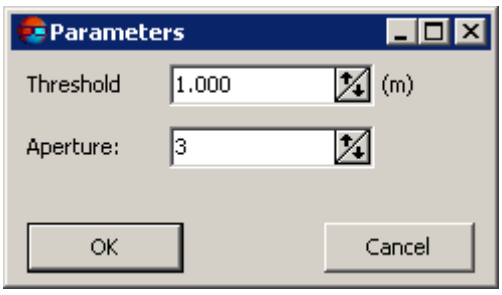
There are two filters available for DEM:

- Median filter
- Smooth filter

8.7.6.1. *Median filter*

The principle of median filter function consists in sequential scanning of the DEM by a *mask* of a given size, defined by the Aperture parameter (measured in DEM grid cells) and replacing Z value in a DEM cell by the mask median value, if the source Z differs more then the **Threshold** from this median value.

To apply the median filter, activate the DEM layer and execute menu command **DEM | Median filter**. Set the parameters and press **OK**.

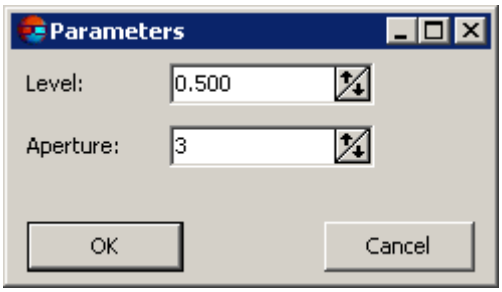


8.7.6.2. *Smooth filter*

To apply the smoothing filter, execute the menu command **DEM | Smooth filter**. The Parameters window opens with the following fields:

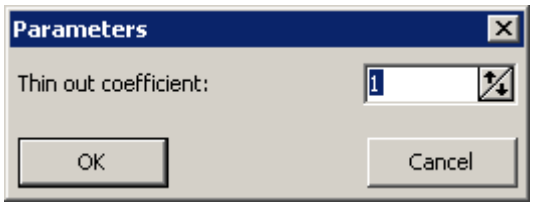
- **Level** – filtration strength in the range from 0 to 1;

- **Aperture** – *mask* size measured in DEM grid cells.



8.7.7. Converting DEM into points

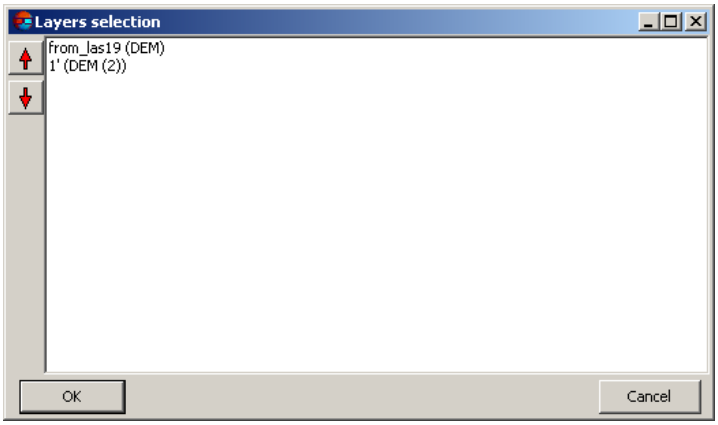
Vector points can be created from a loaded DEM. This is achieved with **DEM | Convert to points** menu command, which opens the Parameters window, containing a single setting – Thin out coefficient, defining number of DEM grid cells used to create each point.
If it is set to 1, a point is created in each DEM cell (no subsampling), so that distance between point is equal to cell size; if set to 2 – in every other cell, etc.



Also points without subsampling may be obtained by exporting the DEM into CSV format (see chapter Export of DEM) and subsequent importing of this file into vector layer. See chapter Import from CSV format.

8.7.8. Difference DEM creating

Using command **DEM | Build difference DEM** one can subtract one DEN from another one and obtain the difference DEM for analysis and investigation. See also DEM's comparing.
Before the command execution the DEMs must be loaded and minuend and subtrahend DEMs selected. On the execution of the command **Layers selection** window will open displaying all DEMs opened.



It is necessary to select two DEMs. Using the arrows put the minuend DEM up (or subtrahend DEM down). Select the DEMs (using **Shift** hotkey) and press **OK**. The window

Save opens – input difference DEM's name and point out its place in the resources. On pressing the button **Save** the DEMs subtraction operation runs.
Switch on the option **Marker Z coordinate follows DEM in active layer** on the tab **DEM** in the Window **Settings** (menu **Service | Settings**). See section DEM settings.

8.7.9. Filling void cells

In most cases, during the automatic DEM creation a number of cells can not be calculated. Such DEM cells (the "holes"), elevation in which is unknown, are called *void cells*.

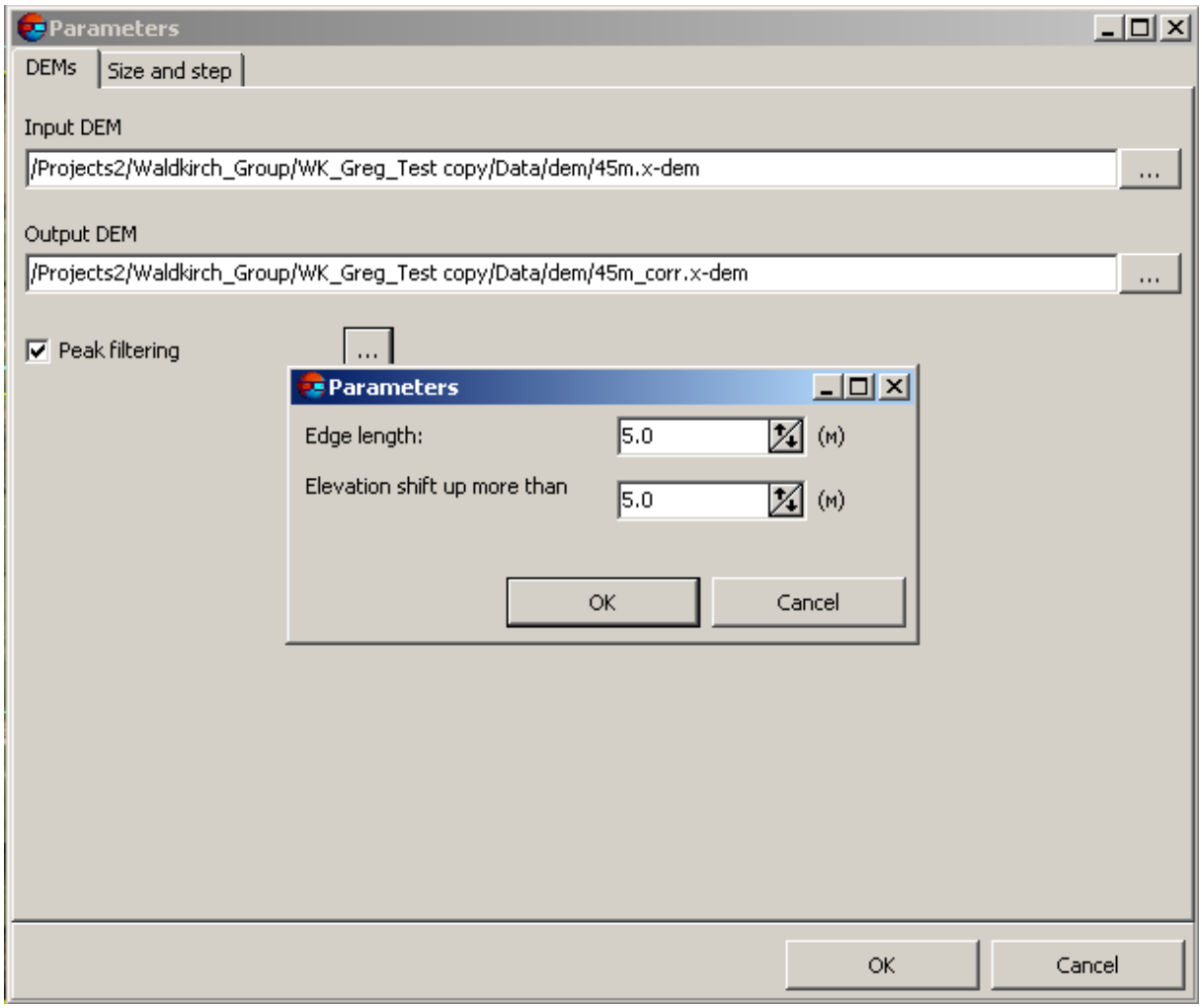
The system provides the possibility to recover DEM void cells. This capability is implemented in the form of the following three methods of filling void cells:

- Linear interpolation
- Smooth interpolation
- By adjacent values

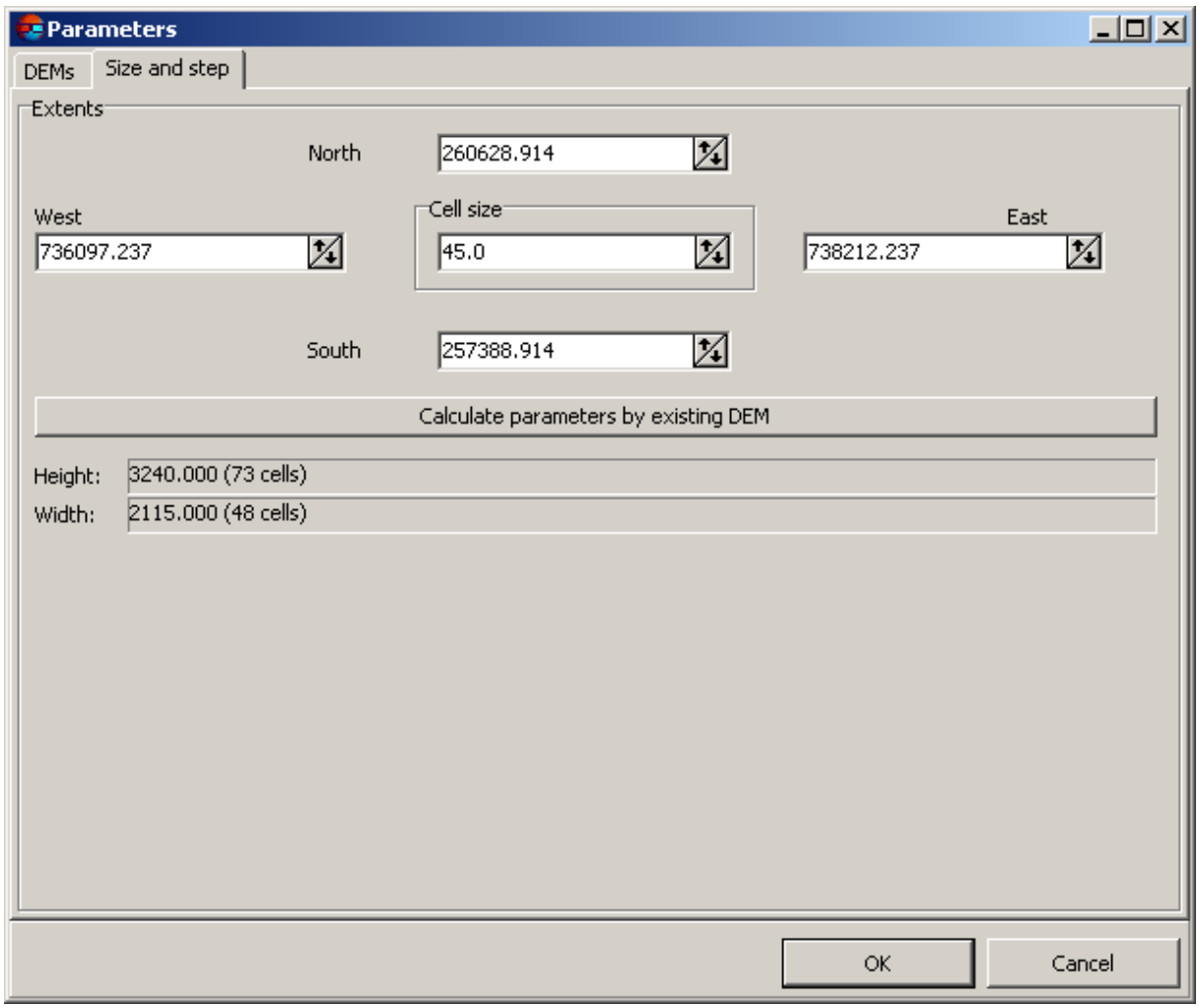
Filling of void cells using the linear interpolation

This method consists in building TIN from DEM and height computation to the void cell by linear interpolation by this TIN. Possibility of boundary changes and DEM cell size for processing, as well as TIN pre-filtering for removal of surges are implementation features of this method in the system.

Use the **DEM | Fill void cells | Linear interpolation** menu command to recover the DEM with void cells by linear interpolation. **Parameters** window opens, which contains two tabs:
• **DEM's** tab - allows specifying the name and location of input / output DEM, as well as setting the parameters for exclusion surges of TIN. It is proposed the name <name of the input DEM> _corr.x-dem for the output DEM and location in the folder containing the input DEM file by default. **Peak filtering** checking allows to obtain the more "smooth" TIN, but it will perform to recomputation the known heights values of non-void cells. Window of additional parameters for searching of surges from TIN to remove - the maximum edge length and the maximum elevation shift up more than - opens by pushing the button.



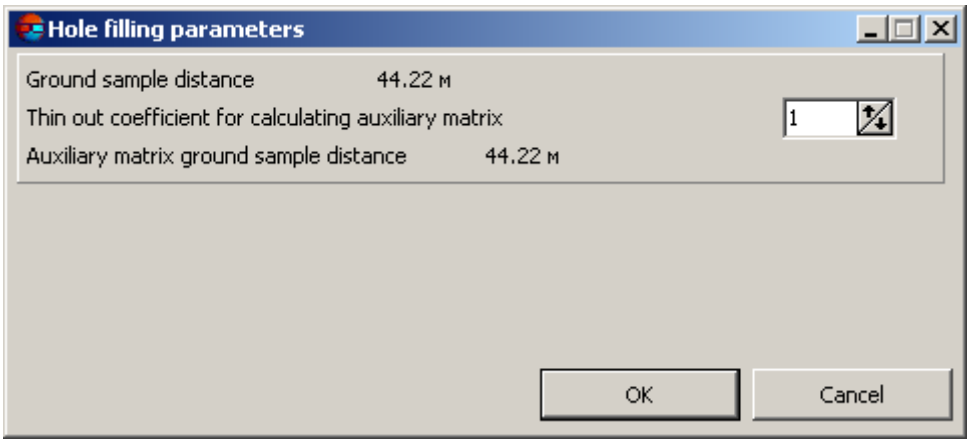
- **Size and step** tab - allows to change the parameters of input DEM. In this tab user can set boundaries and cell size for the input DEM, as well as use the existing DEM.



Filling void cells using the smooth interpolation

This method consists in automatic calculation of the thin out coefficient of input DEM for auxiliary DEM, by which will be calculated elevation values for the void cells by smooth interpolation. This method is recommended for the sufficiently dense DEM. Feature of this method is that the computation of the heights is carried out exclusively for the void cells, i.e the known heights of the input DEM cells are do not change. Output DEM parameters (boundary and cell size) would fully correspond with the input DEM.

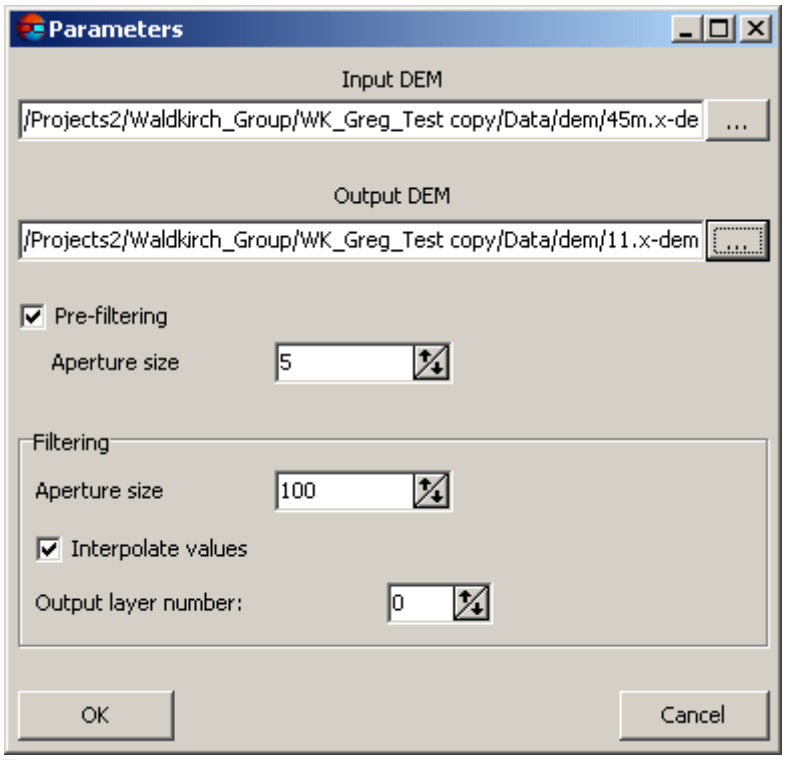
Use the **DEM | Fill void cells | Smooth interpolation** menu command to recover the DEM with void cells by smooth interpolation. **Filling holes in DEM** window opens. The window is used to specify the name and location of input / output DEM, as well as to set the parameters of the auxiliary DEM.
For the output DEM the name <input DEM name> _filled.x-dem and location in the folder, containing the input DEM file are proposed by default. Push the **Parameters** button to open the **Hole filling parameters** window, which displays the GSD of input DEM, as well as automatically calculated values of the auxiliary DEM - thin out coefficient and GSD.



Auxiliary DEM parameters

Filling void cells by adjacent values

Filling void cells with the nearest value method is based on the analysis of cells surrounding the void one. Use the DEM **| Fill void cells | By adjacent values** menu command to recover DEM by this method. **Parameters** window opens, which contains the following settings:



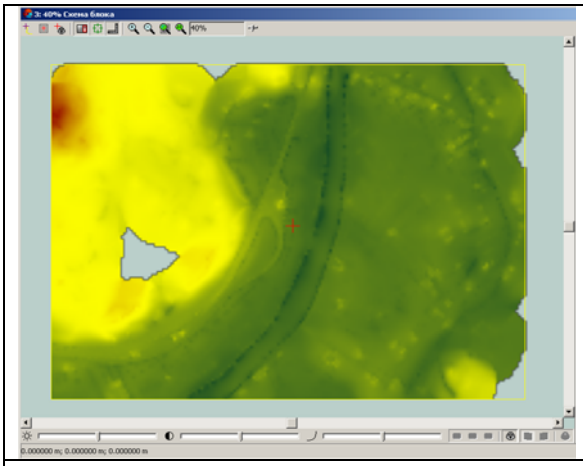
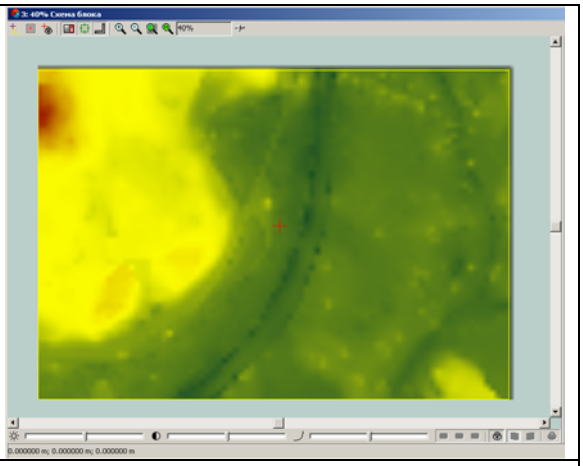
Параметры восстановления пустых ячеек матрицы высот

- **Input DEM** – select from the resources the DEM the NULL values to be restored in.
- **Ouput DEM** – point out the name and place in the profile resources of the output DEM.

- **Pre-filtering** – optional step which comprises values over small neighboring area using main layer of the DEM pyramid. **Aperture size** defines the area. The step allows effective closing the gaps which are the main part of the missing values.

Filtering – mandatory step, to perform which it is necessary to select pyramid layer for output DEM (the main layer is number 0). If the input DEM is too detailed one can thin it out (to set non-zero layer number). The analysis is usually performed with rather large aperture

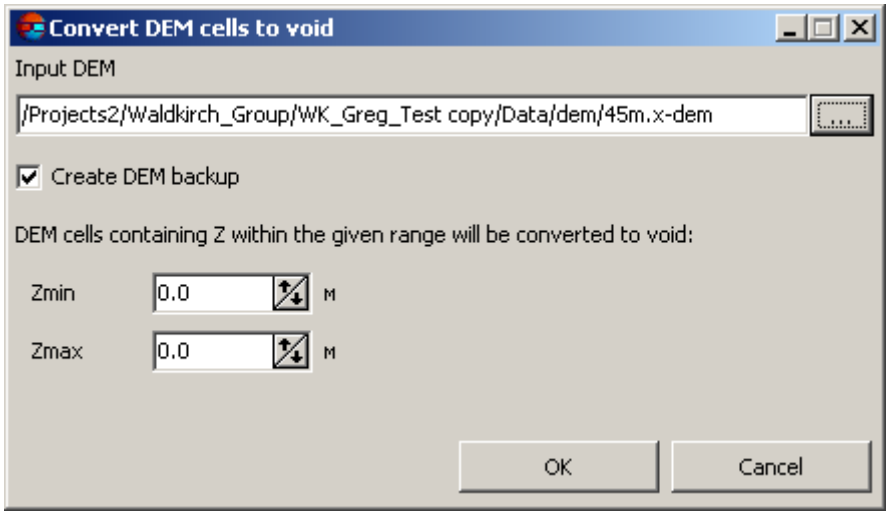
value. It is also possible to use bilinear interpolation for missing values restoring. It will increased the quality as well as processing time.

	
DEM, which contains void cells	Recovered DEM

8.7.10. Conversion cells to void

DEM | Convert cells to void menu command allows to convert the DEM cells with height in a specifed Z-range to the void cells. **Convert Dem cells to void** window opens during the command.

Push the ... button to select the DEM from the resources of active profile for conversion.
Set the Create DEM backup check box to save the input DEM copy, which is saved into the *<input DEM>.backup.x-dem* file of current active profile folder.
Specify the DEM cells for conversion to void. Set its Z-range into the **Zmin**, **Zmax** input box.
Push the **OK** button. All the input DEM cells, which height falls within specified range, are converted to void.

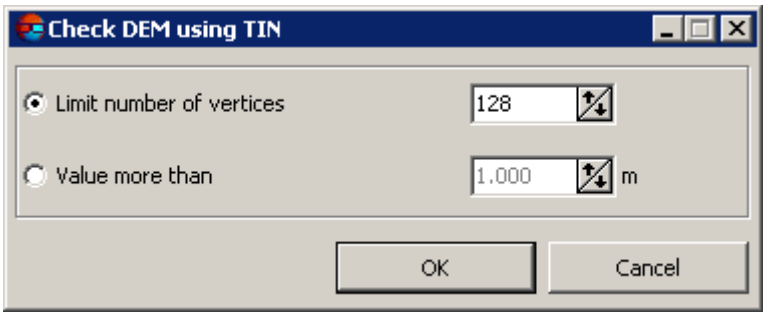


8.8. *DEM accuracy control*

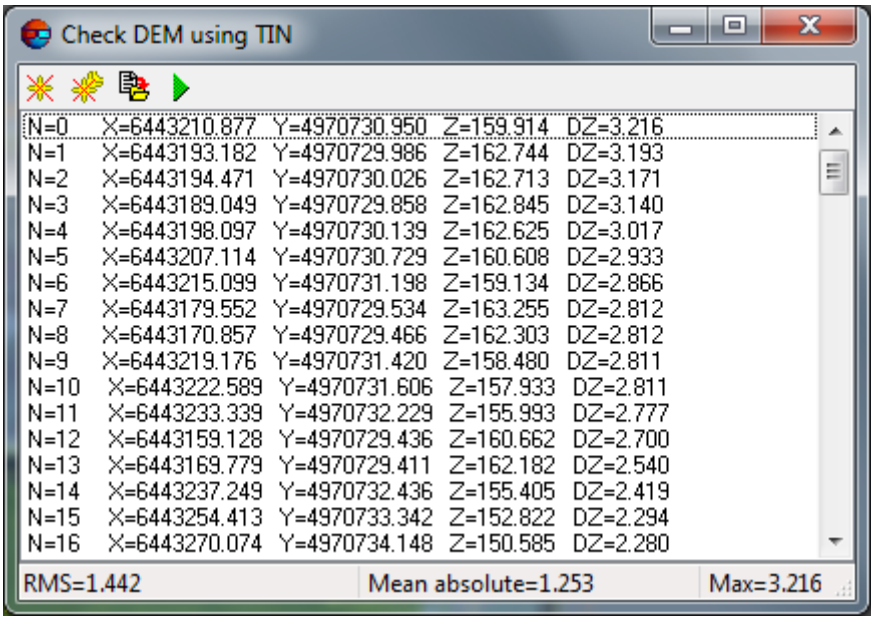
8.8.1. DEM accuracy control by TIN

Menu command **DEM | Check against TIN** is used to calculate a difference in Z coordinate between each source TIN node and the corresponding DEM cell and opens a list of points with the highest difference values. After performing the command the **Check DEM using TIN**

window is opened, allowing to specify maximal number of TIN nodes, where this difference will be calculated.



Operation of DEM accuracy control by TIN is used to “compare” the output DEM and the source TIN. The program calculates a difference in Z coordinate between each TIN vertex and corresponding DEM cell. After the completion of accuracy control operation the **Check DEM using TIN** window is opened, that contains a list with TIN vertices and corresponding deviations.
The list displays X,Y,Z coordinates and value of Z-residual between DEM and TIN for each vertex. The status line shows errors – maximal, mean absolute and RMS.



Results of DEM accuracy control using TIN

Select the vertex in the list to place the marker to appropriate TIN vertex in 2D window to view and edit the TIN. Since each TIN vertex corresponds to picket (point vector object) or node of polyline/polygon of base layers, TIN editing means deletion or editing of the pickets or nodes of vector objects that are the base of TIN. See the chapter TIN editing.

In order to delete vertex of TIN selected in the list, use the **Delete** key or the button. You can also delete all vertices with the residuals more or equals the selected one using the button.

Thus, the operation of DEM accuracy control is one more tool of searching of erroneous TIN vertices – spikes.

Once insufficient vertices are deleted from the list you can re-build the DEM immediately by pushing the button .

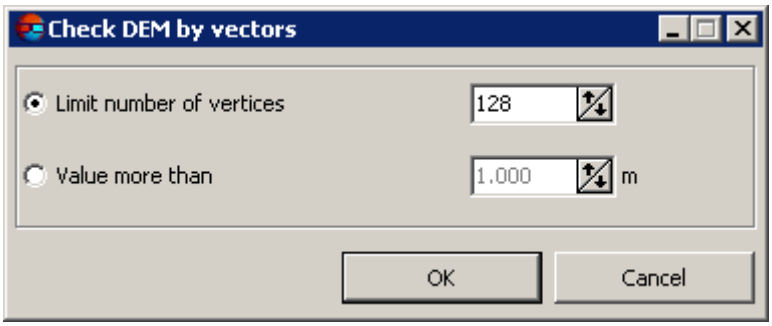
You can save the results of DEM accuracy control (the list of vertices with their coordinates) to the text file by clicking the button in this window.

Erroneous vertices list includes those located on all stereopairs of current project.

8.8.2. DEM accuracy control by vector objects

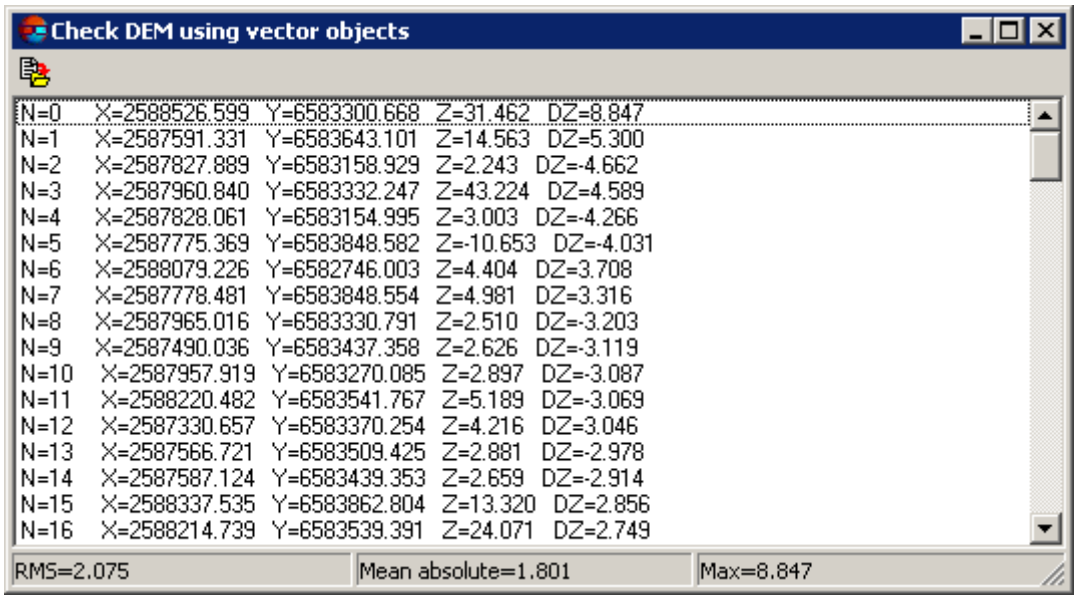
In some cases may be necessary to compare DEM with vector objects, which are not a source data for the DEM creation, i.e. they are not base layers for TIN building, that used in turn for the DEM creation, or they are not regular or irregular pickets used for the DEM creation. The DEM may be also obtained from external sources and you may want to estimate its accuracy using available vector objects. Or vice versa, you may want to estimate vector objects using “reference” DEM.

During comparing ”vectors” with DEM Z-coordinate of pickets or vector objects nodes are comparing with heights of corresponding DEM points. In order to perform such a comparing select menu command **DEM | Check against vector objects**.
The parameters window opens:



- **Limit number of vertices** – only given number of points with maximum Z deviations is shown in the list
- **Value more than** – the list will contain all points and vertices, deviation of which from Z is larger than the given value.

When the calculation process is complete, window with the list opens, containing coordinates of points/vertices and their deviation from DEM (DZ).
The status bar shows maximum, mean absolute Z deviation and its RMS.

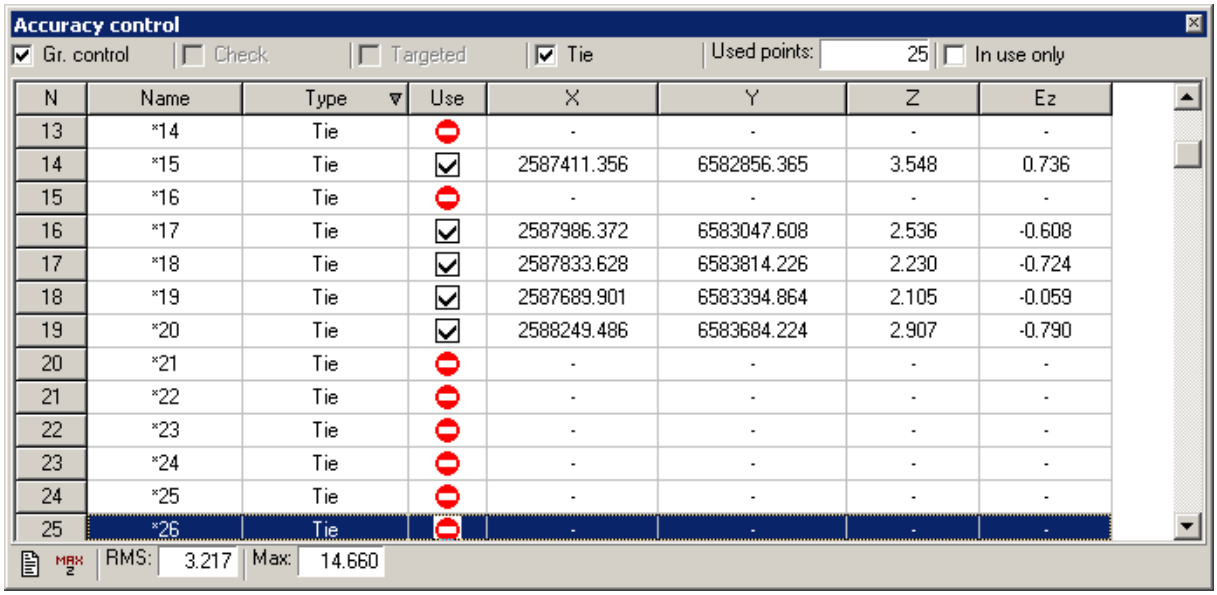


When a point is selected in the list, marker in the 2D window is moved to its coordinates.

You can save the results of DEM accuracy control (the list of vertices with their coordinates) to the text file by clicking the button in this window.

8.8.3. DEM accuracy control by triangulation points

If triangulation points have not been used at some point of DEM creation, it is feasible analyzing the DEM deviation by Z from the triangulation points. The menu command **DEM | Check against adjustment points** is used for this purpose. There is no need to load the points into a vector layer. After the calculation process is complete, the **Accuracy control** window opens containing all points (ground controls/check/tie) obtained on the block measurement stage (see **Aerial Triangulation** user manual).



Accuracy control by triangulation points

The top panel of the window is used to control visibility of different kinds of points in the list and allows displaying **ground control**, **check**, **tie** and/or **targeted** points, as well as used

(falling within the DEM border, having the Z discrepancy computed) only points. Also the quantity of used points is displayed in the panel.
Each point in the list has the following data in the corresponding columns:

- point **Name** and **Type**;
- whether the point is **used** in DEM Z discrepancy control (- not used, - used);
- adjusted coordinates of the point (**X**,**Y** and **Z**);
- DEM discrepancy by Z (**Ez**).

Besides, the RMS and Maximum error by Z is displayed in the bottom panel.
The button is used to find and select in the list the point with maximum discrepancy.
The button is used to open and print the detailed accuracy report.
The **Name, Type, Use** and **Ez** columns may be sorted in ascending or descending order (by clicking the column title).

8.8.4. Void cells searching

Feature of void cells searching in DEM (“holes”) is provided in the system to analize the DEM building quality.
Select **DEM | Search for void cells** menu command to start search for void cells (“holes”) in DEM. Then the window containing the **Search for DEM fragments** parameter is opened.
Push the **OK** button. The operation of searching for void cells starts. Once the search is completed you will get opened the list of found void cells with its X, Y-coordinates centers. When the element is selected in the list, the marker moves to appropriate DEM cell. You can save the list to the text file by clicking the button.

The following features to edit DEM with void cells are provided:

- There are three methods of filling void cells (see chapter Filling void cells);
- Flattening fragment;
- Edition of vector objects and DEM rebuilding, in the case if the source vector layers are used during the DEM building.

8.8.5. DEM’s comparing

To perform a comparing of two DEM’s for the same territory in order to find deviations by Z-axis between them, use menu command **DEM | Compare DEM’s**. After selecting the command the **Select layer** window is opened, that is used to choose two DEM’s. You can specify a limit of points number to be displayed in the list. The list displays first of all points with maximal difference in Z values.

8.9. *LIDAR data loading*

See **Using the LIDAR data in the system** User Manual.

9. Import-Export

9.1. *Import vector objects*

For importing and exporting vector objects, submenus **Vectors | Import** and **Vectors | Export** are used respectively.

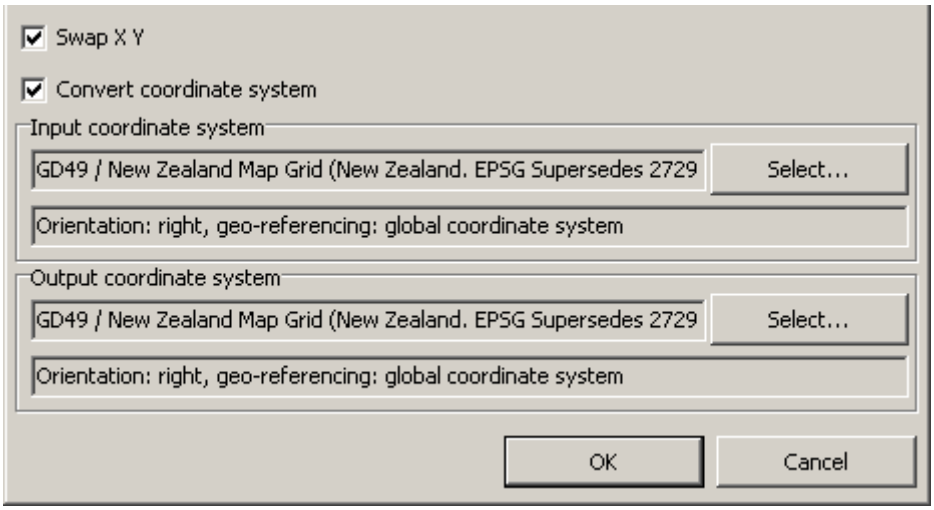
During import of the objects with classifier from external exchange formats, the objects are linked to current Code table (see the chapter Classifier). For this the information, stored in main imported file or in DBF-file accompanying main imported file, is used.

Importing any vector format begins with the system standard open file dialog for selecting one or more files to import. Then the import parameters dialog is shown for settings parameters of imported coordinates conversion, and parameters specific to the format being imported. Finally, depending on one or several files are imported, whether there is an open vector layer on PHOTOMOD, etc. there may be shown a list of options, containing a subset of the following set of actions, in similar fashion to loading vector data from resources:

- **Load into one layer** – (if there is no loaded vector layer) create new vector layer and load all selected files into it
- **Create separate layer for each file** – create a new separate layer for each of the files being imported
- **Batch import** – if this option is selected, on the next step a folder in the resources system must be selected, then in the process of import each of the selected files is loaded, saved to the given folder, and closed. This way can be used to load large amount of data split into lots of moderately-sized files, while the total amount of data or the processing workflow makes it undesirable or impossible to load into single PHOTOMOD layer
- **Clear layer and continue loading into it** – when there is loaded vector layer, purge it (with saving confirmation if there are unsaved data) and load all imported files into it
- **Load all into one new layer** – (when there is loaded vector layer) create one new vector layer and import all selected files into it
- **Load all layers into layer without unloading existing data** – when there is a vector layer with data, load all files into it without purging existing data.

9.1.1. Coordinates conversion during import

For all import formats, there are options during import process to swap X and Y plane coordinates, of imported points, and convert vector data from one coordinate system to another. A standard panel at the bottom of all import parameter dialogs is used to control this process, shown on the illustration:


Swap X Y
Convert coordinate system
Input coordinate system
GD49 / New Zealand Map Grid (New Zealand. EPSG Supersedes 2729
Select...
Orientation: right, geo-referencing: global coordinate system
Output coordinate system
GD49 / New Zealand Map Grid (New Zealand. EPSG Supersedes 2729
Select...
Orientation: right, geo-referencing: global coordinate system
OK
Cancel

When "**Swap X Y**" option is checked, plane coordinates of input points are swapped.
When "**Convert coordinate system**" option is checked, input and output coordinate systems are assigned in a fashion common to all PHOTOMOD system – the "Select" button provides for selecting a coordinate system from preinstalled database, from file or from resource. The default setting for input and output coordinate systems is the coordinate system of the loaded project.

Note that if both options ("**Swap X Y**" and "**Convert coordinate system**") are checked, first of all plane coordinates of points are swapped and then the result is converted between coordinate systems.

9.1.2. Importing attributes

In order to use classifier and import attribute values for the loaded vector data, the target layer must have been assigned a classifier beforehand (i.e. created with **Vectors | Create layer with classifier** command or loaded from resource containing data with classifier). During import into layer without classifier, objects semantics may be partially or totally discarded.
When objects attached to an external classifier are imported, they are linked to currently loaded classifier. For correct linking, information is used which is contained in the primary file being imported or in accompanying DBF file.
The latter DBF file contains a database table in dBASE format, containing rows (records) and columns (fields).
Database fields describe attribute data. In particular, first 4 fields of DBF file, created when exporting vector data to external format, contain the following information:

- NAME – unique **object name**, which is stored in an attribute with fixed name "Name", if this attribute is defined in classifier
- CODE – classifier **code**, which the object is assigned to
- CODENAME – classifier **code name**, which the object is assigned to
- LAYER – classifier **layer name**, which the object code is stored in.

Fifth and further fields contain main and additional attributes of vector objects.

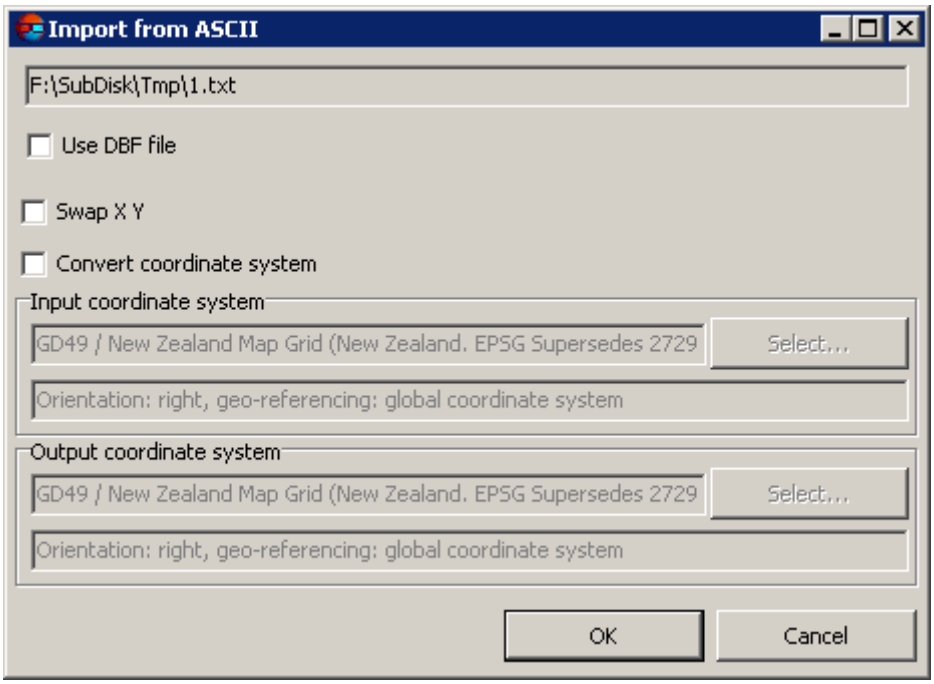
The following schematic example illustrates the DBF file structure used by PHOTOMOD. Let there be several vector objects:
Object **A** has attributes - **a, b, c**
Object **B** has attributes - **b, c**
Object **C** has attributes - **b, k**
This set would be saved to and loaded from the following DBF table:

	NAME	CODE	CODENAME	LAYER	attributes			
					a	b	c	k
A	☑	☑	☑	☑	☑	☑	☑	
B	☑	☑	☑	☑		☑	☑	
C	☑	☑	☑	☑		☑		☑

Some formats (e.g. ASCII-A) store objects attributes not in a separate DBF file but inside the main file. In such case attributes of vectors objects during import would be loaded and linked to classifier using main file only.
The following procedure is used to link imported objects to existing classifier: code, code name and object type (point, polyline or polygon) are read from DBF or main file, and code entry with such characteristics is checked to exist in the classifier. If the entry exists, the object is linked to it. Otherwise, a new layer is created, and a code entry possessing the said code, code name and type is created, with imported object linked to it.

9.1.3. Import from ASCII format.

To import data from PHOTOMOD ASCII format (see format description below), execute menu command **Vectors | Import** | **ASCII**. Files in this format have ".txt" extension. "Simple" ASCII format is designed to store only vector geometry data without semantics, attributes data can be stored separately in a DBF files. The format does not imply storing 2D objects.
The import parameters dialog for this format looks as follows:



If a DBF file is present, vector objects are linked to classifier in a manner described in Importing attributes section. In order for this to work, DBF file must be placed in the same folder as .txt file and have the same name and extension .dbf, and the **Use DBF file** option must be checked in parameters dialog.
If no DBF file is used during import, entries for codes stored in the ASCII file itself will be created in the classifier root layer, and imported objects will be linked to these code entries.

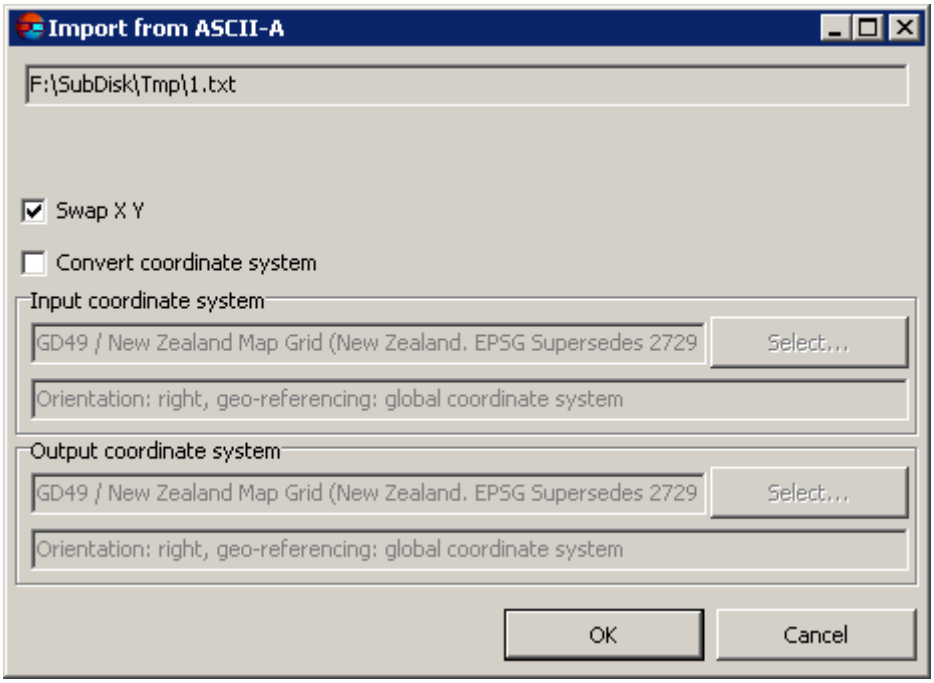
9.1.3.1. *ASCII format description*

This format represents each vector object as a sequence of points defined by 3D coordinates.
Formally the file consists of records of variable length separated by the "*" (asterisk) symbol. Each record defines an object which may be either point or polyline.
The record consists of object code and a sequence of lines with comma-separated 3D coordinates of points. Example of such file:

```
Road
1234.67,4567.67,565.453
1245.6,7439.570,860.958
*
Point
1257.85,2198.76,459.56
*
```

9.1.4. Import from ASCII-A format

To import data from PHOTOMOD ASCII-A format (see format description below), execute menu command **Vectors | Import** | **ASCII-A**. Files in this format have ".txt" extension. ASCII-A format contains objects attributes along with geometry information, thus objects attributes are loaded directly from the ASCII-A file. The format does not imply storing 2D objects. The import parameters dialog for this format contains only standard coordinate conversion settings, there are no format-specific options.



9.1.4.1. *ASCII-A format description*

ASCII-A format, aside from coordinates of vector objects, contains information about objects type, layer numbers, as well as attributes names and values. This format is an extension of "simple" ASCII file (see section ASCII format description), with object definition complemented by lines describing type and attributes of the object.
See notes below describing the following example:

```
L 101 1 15 7
OBJECT_NAME=Railway
OBJECT_COLOR=3
OBJECT_SYMBOL=#
OBJECT_SIZE=5.5
WIDTH=7.2
ROADNAME=London-Paris
545566.505,473671.817,77.850
545715.103,473656.072,78.310
545782.001,473567.393,78.156
545860.428,473463.139,77.974
545847.506,473339.305,77.380
545795.032,473249.288,76.795
545517.126,473365.500,76.318
545269.605,473463.426,75.869
*
```

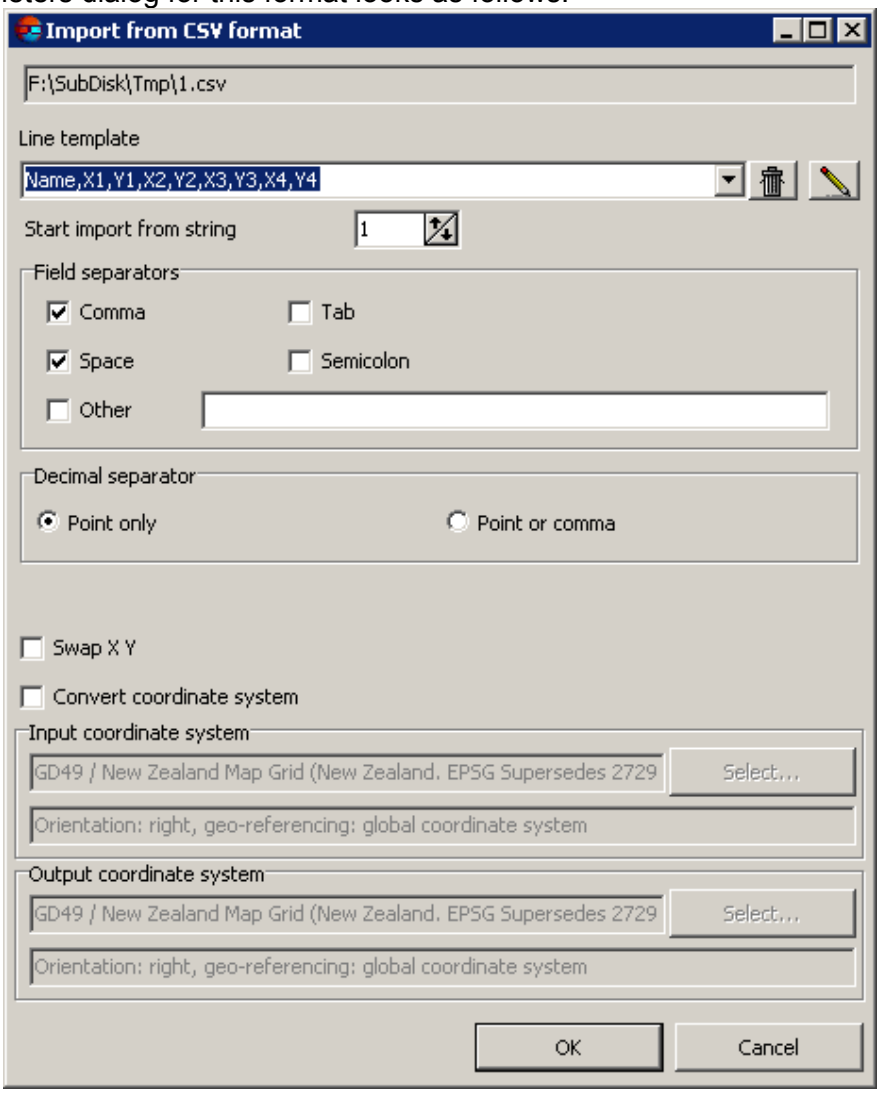
The first line of the object definition contains fields:
`[Type] [Code] [Layer] [N1] [N2]`,
where:
• **Type** – symbol defining the object type (L, P, C);
• **Code** – object code;
• **Layer** – layer number;
• **N1** – total number of lines taken by object definition;
• **N2** – number of lines taken by attributes listing.
Next lines contain attributes description in the form of:
`Name=Value`
There are for fixed attributes:

• **OBJECT_NAME** – object name;
• **OBJECT_COLOR** – object color code;
• **OBJECT_SYMBOL** – object symbol;
• **OBJECT_SIZE** – object size.

These attributes got to the properties **Name**, **Color**, **Symbol** and **Size** of code entry.
Attribute lines are followed by lines with coordinates of object points, in the same form as for the ASCII format (see section ASCII format description). Object definitions are separated by an "*" (asterisk) symbol.

9.1.5. Import from CSV format

CSV format is a rather flexible exchange data format to one degree or another supported by vast variety of programs of different profiles, and also easily human-editable. Due to this fact it can be used as an exchange format in case when specialized geospatial formats cannot be used for some reason. In particular, CSV is often used as an exchange format for orthophoto sheet frames specification data. CSV import function in PHOTOMOD is primarily oriented to the latter use, but may be as well used for a broad spectrum of other tasks.
To import data from CSV format, execute menu command **Vectors | Import** | **CSV**. The import parameters dialog for this format looks as follows:



The **Line template** field is used for defining the sequence of fields contained in each line of the imported CSV file. The following field types may be listed, separated by commas:

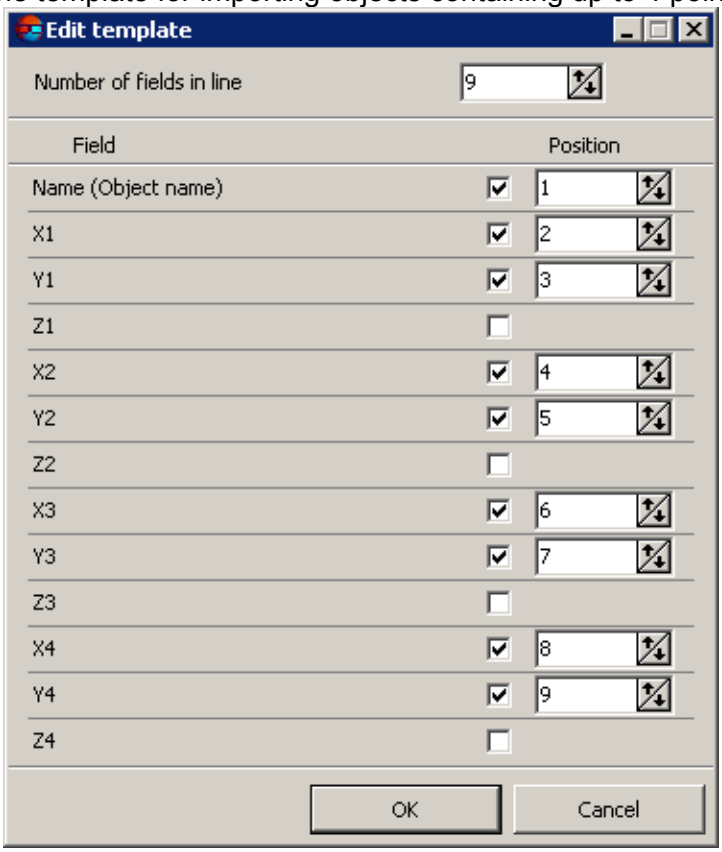
- **Name** – object name

- **Xn**, **Yn**, **Zn**, where **n** is an integer number starting from 1 – coordinates of the object point no. **n** respectively
- * - field to bypass

All objects stored in the file must follow the same template, i.e. each CSV line must contain the same number of fields, corresponding to number of fields in the template. Lines not matching the template are ignored. For all points, 2 (for 2D objects import) or 3 coordinates must be available.

When the "**OK**" button is pressed, active template is saved, and when the command is called next time, added to the drop down list containing history of templates. The "**Clear list**" button may be used to clear this history list.

The "**Build template**" button calls the CSV template editor window, which can be used to visually construct the template for importing objects containing up to 4 points:



This window allows setting total number of fields in line and position of each field. When the "OK" button is pressed, the "**Line template**" field in import parameters dialog receives the resulting template.

The default template:

Name, X1, Y1, X2, Y2, X3, Y3, X3, Y4

Matches lines containing for each object its name and 2D coordinates of 4 points, i.e. standard record for orthophoto sheet frame specification.

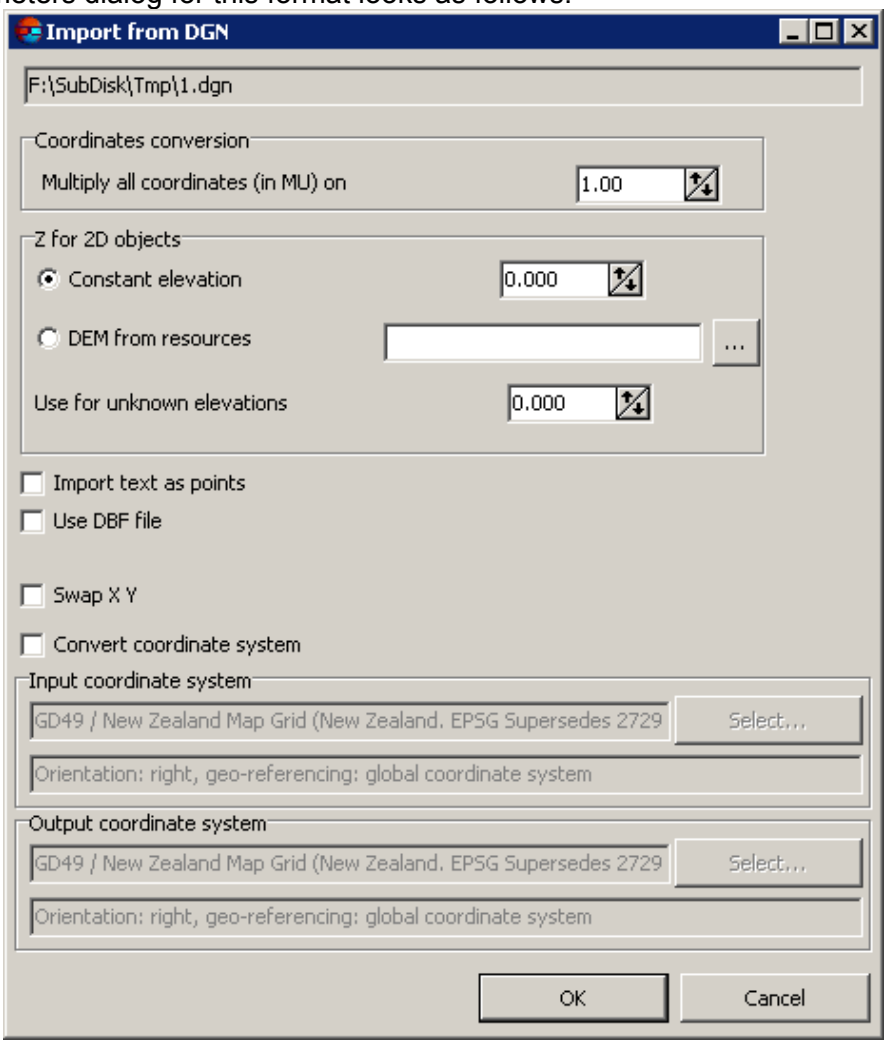
The "**Field separators**" group is used to define list of symbols separating fields in the CSV file.

The "**Decimal separator**" option defines whether only decimal point may be used to separate fractional part of a number from the integer part, or there may be either point or comma used (situation incidental to files created by programs respecting operating system locale setting in Russia and several other countries). In the latter case a comma cannot be used as a field separator, or create objects would have erroneous coordinates or the import process may fail at all.

The field defined as an object name is assigned to attribute "Name" of the created objects.

9.1.6. Import from DGN format

PHOTOMOD supports import from DGN V7 format (also known as ISFF DGN or Intergraph DGN), used in MicroStation J / 7. **DGN V8 format, used by default in MicroStation 8 and later, is not supported.** To import DGN data from such system, it is necessary to save DGN as V7 by the means of MicroStation before loading it into PHOTOMOD.
To import data from DGN format execute menu command **Vectors | Import | DGN**. The import parameters dialog for this format looks as follows:



The "**Coordinates conversion**" field makes it possible to multiply all coordinates in the file by the same factor during import (1.0 by default – i.e. no change).
2D DGN objects are converted to 3D during import. The way to obtain Z coordinates for object points may be chosen from the following options (in "**Z for 2D objects**" pane):

- **Constant elevation** – all objects are assigned the same Z value
- **DEM from resources** – allows to select a resource with DEM to peek Z values from
- **Use for unknown elevations** field defines the Z value to be used for points falling into DEM void cells (cells with no elevation assigned)

When "**Use DBF file**" option is checked and there exists corresponding DBF file in the folder with DGN file, imported vector objects are linked to classifier as described in section Importing attributes.

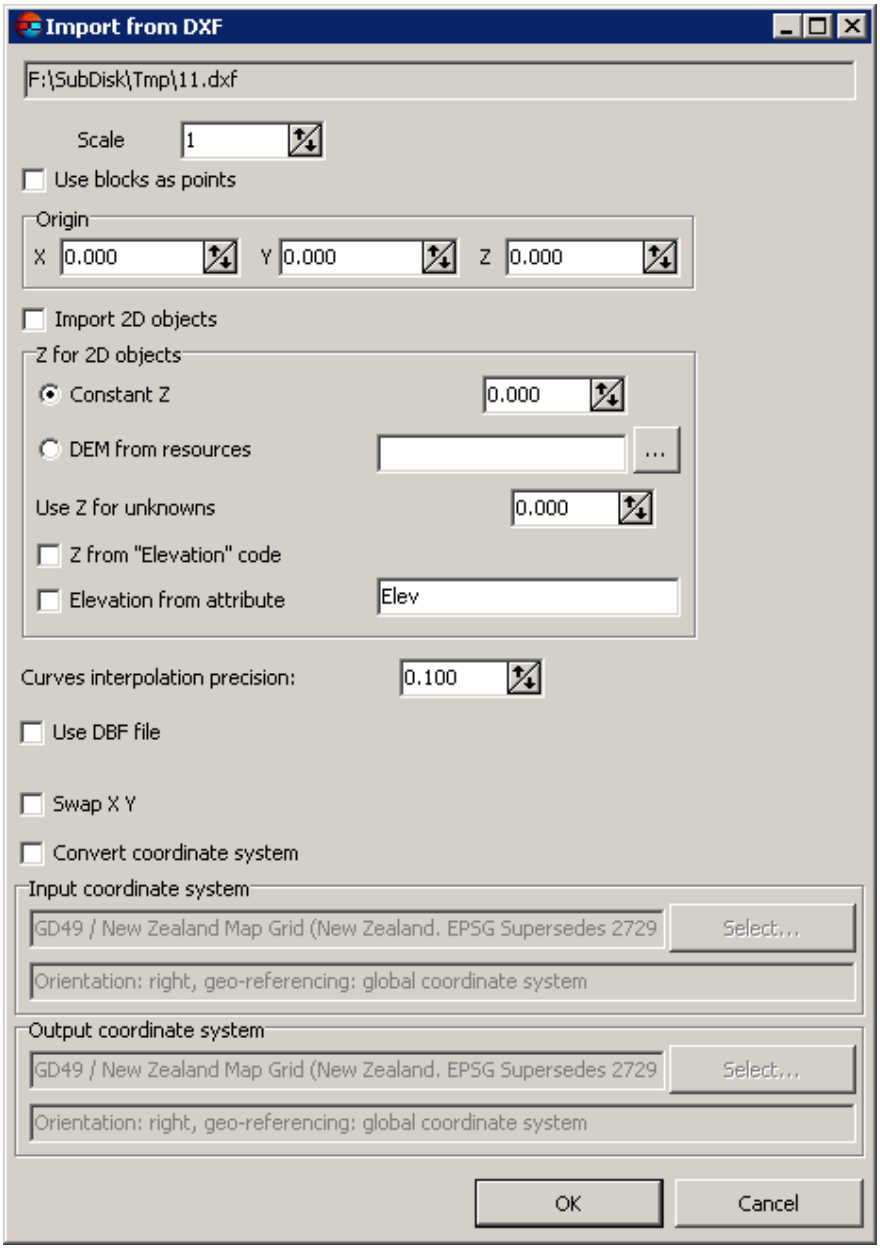
If no DBF file is used for import, objects are linked to classifier in the following manner: in the classifier root layer layers and code entries corresponding to layers and codes of DGN file are created, and objects are linked to these code entries.

If "**Import text as points**" is set, DGN text objects are imported as point objects with "Label" attribute containing text of the source object (see section *Creating labels*).

Some peculiarities of the DGN V7 format are described in section Export to DGN format.

9.1.7. Import from DXF format

To import data from DXF format (an AutoCAD exchange format) execute menu command **Vectors | Import | DXF.**
DXF R14 (AutoCAD R14 format) is supported. Import of newer versions of DXF is not supported.
The import parameters dialog for this format looks as follows:



The **"Scale"** field allows multiplying all coordinate values in file by the same factor (1.0 by default – i.e. no change).
The **"Use blocks as points"** options allows importing origins of AutoCAD blocks as point objects.

The **"Origin"** panel is used to set origin of coordinates during import. Default values – (0.0, 0.0, 0.0).
The "**Import 2D objects**" option defines whether 2D objects are imported and converted to 3D. The way to obtain Z coordinates for object points may be chosen from the following options (in "**Z for 2D objects**" pane):

- **Constant elevation** – all objects are assigned the same Z value
- **DEM from resources** – allows to select a resource with DEM to peek Z values from
- **Use for unknown elevations** field defines the Z value to be used for points falling into DEM void cells (cells with no elevation assigned)
- **Z from "Elevation" code** instructs the system to import Z from the Elevation attribute of DXF file
- **Elevation from attribute** – use an attribute in DXF file (name of which is entered in the field) to obtain Z from.

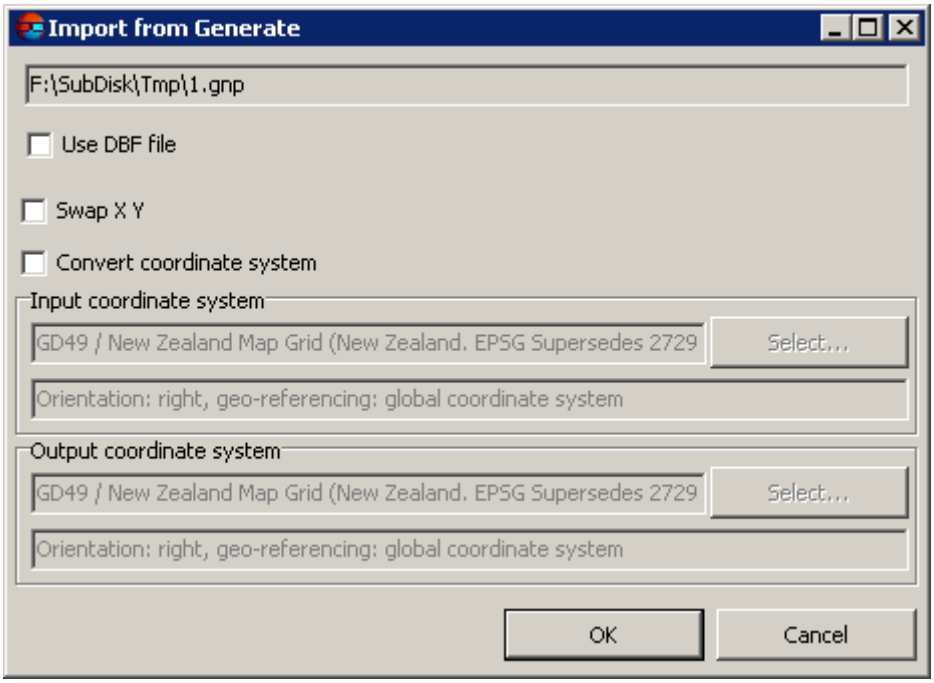
"**Curves interpolation precision** ": when objects containing smooth curves are imported, the latter are approximated by polylines. Value in this field defines precision (maximum distance from source curve to the polyline), thus influencing the number of nodes created.

When "**Use DBF file**" option is checked and there exists corresponding DBF file in the folder with DXF file, imported vector objects are linked to classifier as described in section Importing attributes.

If no DBF file is used for import, objects are linked to classifier in the following manner: in the classifier root layer layers and code entries corresponding to layers and codes of DXF file are created, and objects are linked to these code entries.

9.1.8. Import from Generate format

To import data from ArcINFO Generate format execute menu command **Vectors | Import | Generate.** Common extensions for this type of files are .gnp, .pnt for files with points and .gnl, .lin for files with polylines.
The import parameters dialog for this format looks as follows:

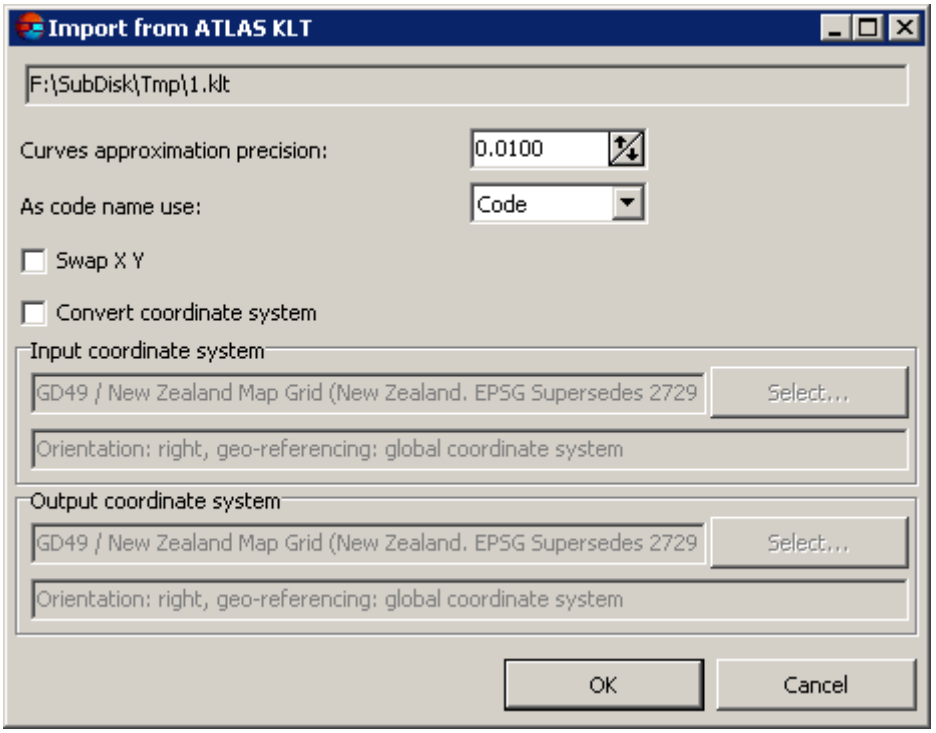


If no DBF file is used during import, i.e. the only additional information about object is its type (point, polyline, polygon) from the main file, the objects are linked to codes created in the root layer of classifier for the type of objects in the file (either points or polylines, or polygons).

When "**Use DBF file**" option is checked and there exists corresponding DBF file in the folder with Generate file, imported vector objects are linked to classifier as described in section Importing attributes.

9.1.9. Import from ATLAS KLT format

To import data from ATLAS KLT format execute menu command **Vectors | Import | ATLAS KLT.**
The import parameters dialog for this format looks as follows:



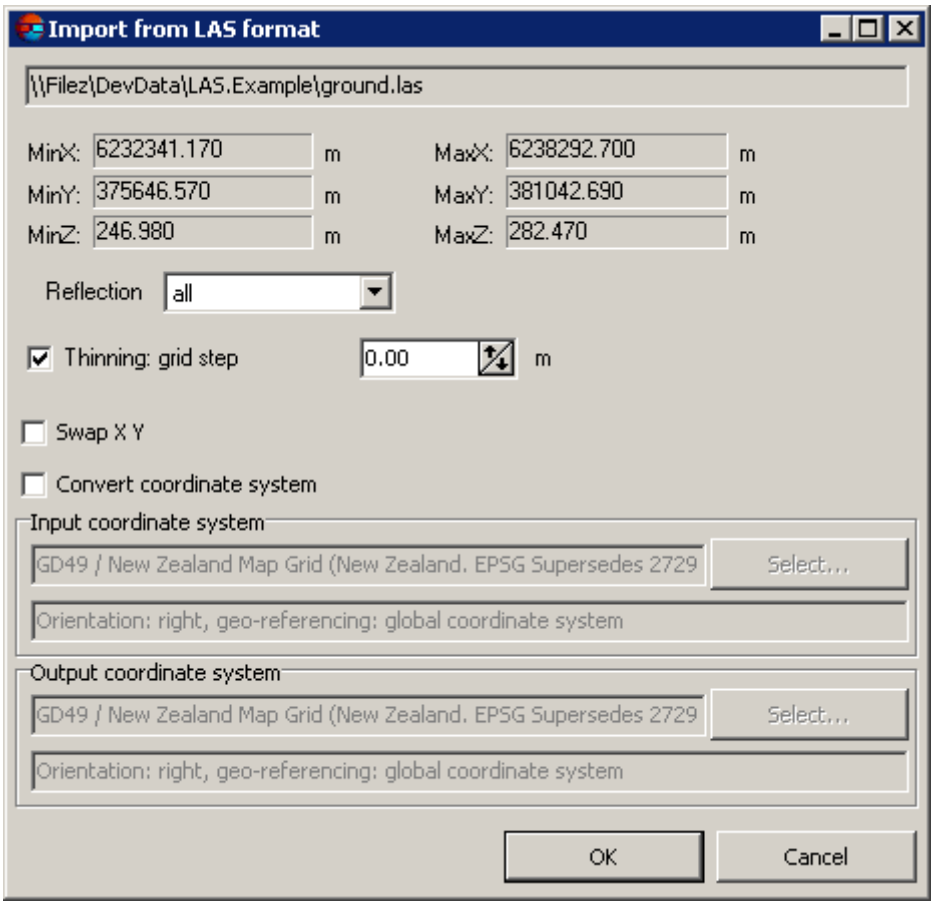
"Curves approximation precision" parameter influences import of arc and circle objects: the higher the precision (smaller value) the more nodes are added to polylines approximating these curves.

When "**Use DBF file**" option is checked and there exists corresponding DBF file in the folder with KLT file, imported vector objects are linked to classifier as described in section Importing attributes.

If no DBF file is used for import, objects are linked to classifier in the following manner: in the classifier root layer layers and code entries corresponding to layers and codes of KLT file are created, and objects are linked to these code entries.

9.1.10. Import from LAS format

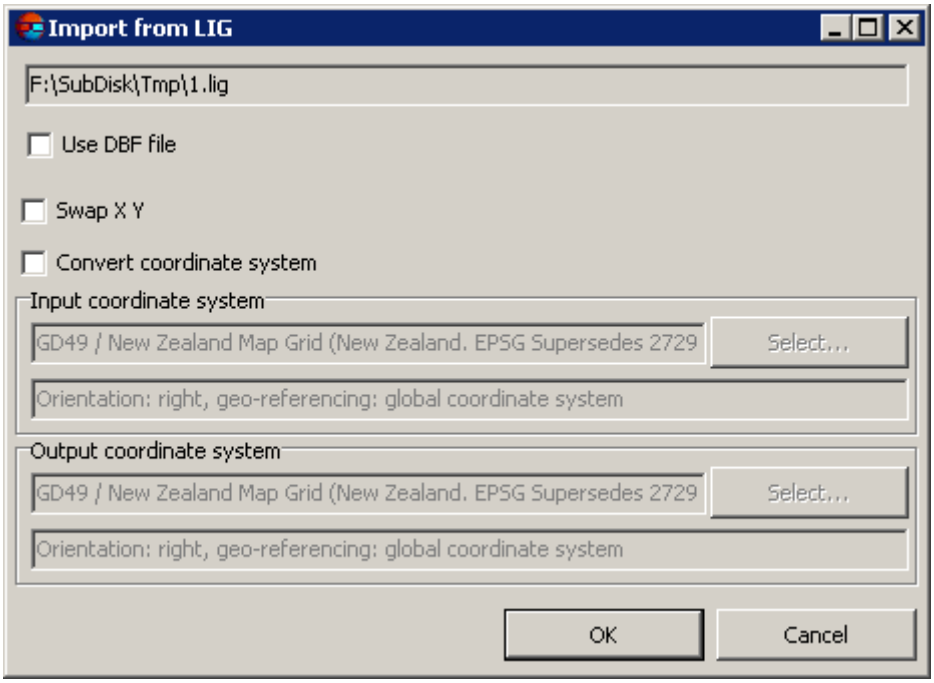
To import data from LIDAR LAS format execute menu command **Vectors | Import | LAS.**
The import parameters dialog for this format looks as follows:



MinX … **MaxZ** labels show the range of coordinates of points contained in the LAS file to import.
The "**Reflection**" drop-down list is used to specify which reflection numbers should be loaded. "**All**" is chosen by default, i.e. import all points contained in the file.
The "**Thinning**" option allows lessening of the number of imported points by thinning them to the given density with minimum distance between points specified in the "**grid step**" field.

9.1.11. Import from LIG format

To import data from LIG format execute menu command **Vectors | Import | LIG.**
The import parameters dialog for this format looks as follows:

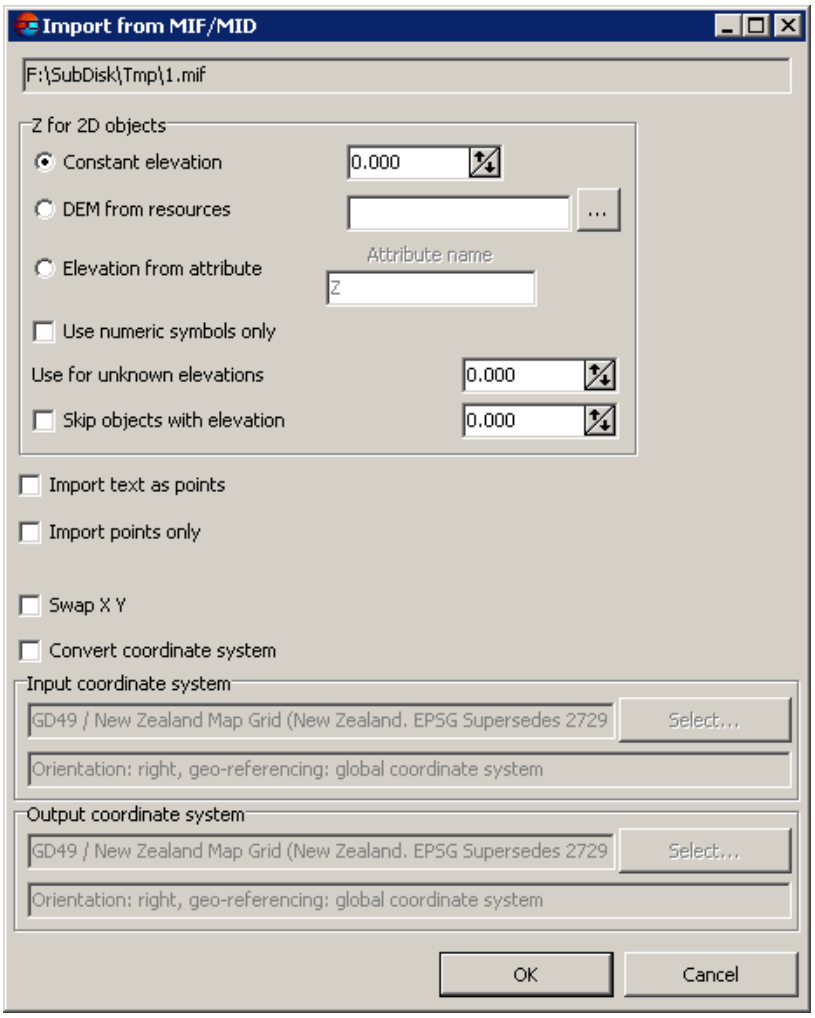


When "**Use DBF file**" option is checked and there exists corresponding DBF file in the folder with LIG file, imported vector objects are linked to classifier as described in section Importing attributes.

If no DBF file is used for import, objects are linked to classifier in the following manner: in the classifier root layer layers and code entries corresponding to layers and codes of LIG file are created, and objects are linked to these code entries.

9.1.12. Import from MIF/MID format

To import data from MapInfo MIF/MID format execute menu command **Vectors | Import | MIF/MID.** In the open file dialog, files with extension .mif should be selected.
The import parameters dialog for this format looks as follows:



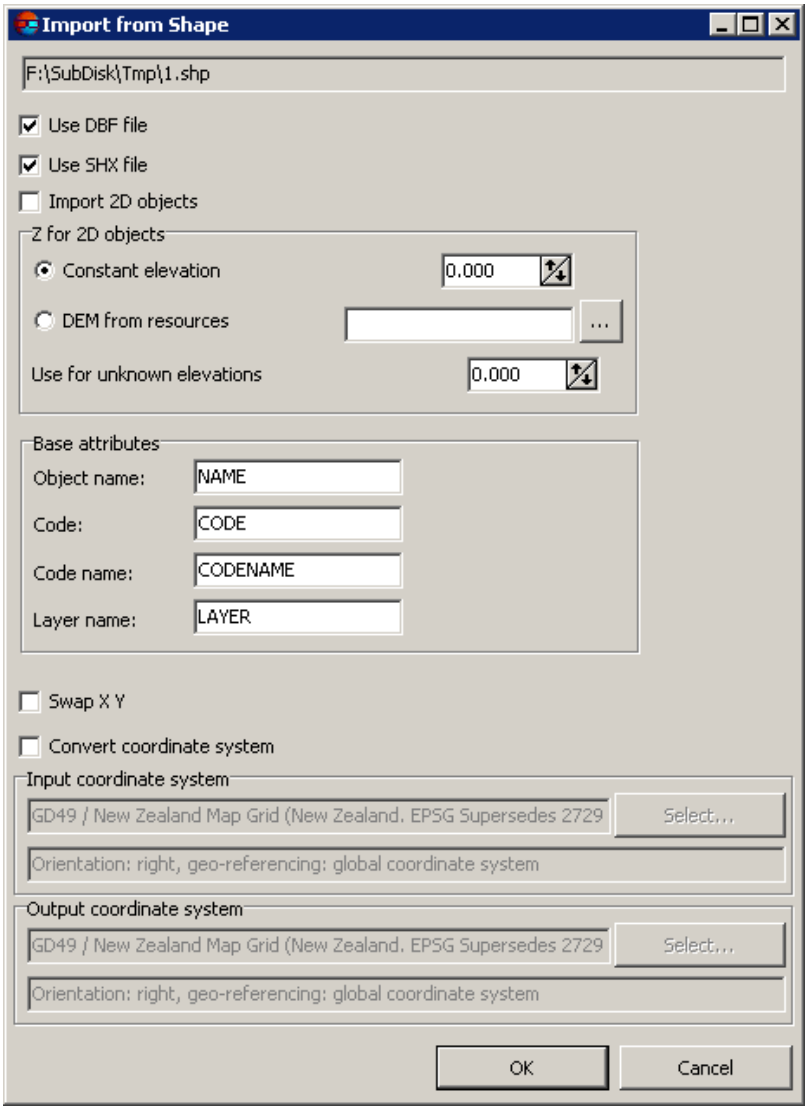
MIF/MID format is essentially 2D, but Z coordinates may be still obtained from it, a single Z value for all points of an object, which is commonly used to import contours. The general source of Z coordinates is defined in "**Z for 2D objects**" panel:

- **Constant elevation** – all objects are assigned the same Z value
- **DEM from resources** – allows to select a resource with DEM to peek Z values from
- **Elevation from attribute** – if Z coordinates are stored in an attribute in MID file as mentioned above, the name of this attribute must be entered in this field in order to use these elevation values.
- **Use numeric symbols only** – if this is checked, values of the attribute specified in the **Elevation from attribute** field are filtered so that only digits, point, comma and minus sign are left, and then comma if present is treated as a decimal point (this is essential working in Russian and some other locales using comma as the decimal separator). Thus, it becomes possible to read Z from the attribute string `"elev:90.6:note::"`. Still in case the Z attribute is supposed to be of proper floating point format, this options should be unchecked to avoid some ambiguousness brought by such kind of parsing.
- **Use for unknown elevations** – Z coordinates which could not be obtained with preceding methods are assigned the value entered in this field
- **Skip objects with elevation** – when an attribute is used as a source of elevation data, if it is equal to the value entered in this field, then the corresponding object is considered invalid and is not imported.

If "**Import text as points**" is set, MIF/MID text objects are imported as point objects with "Label" attribute containing text of the source object (see section *Creating labels*).

9.1.13. Import from Shape format

To import data from ArcINFO Shape format execute menu command **Vectors | Import | Shape.**
Files in this format usually have .shp extension. One file may contain objects of a single type (either points or polylines, or polygons).
The import parameters dialog for this format looks as follows:



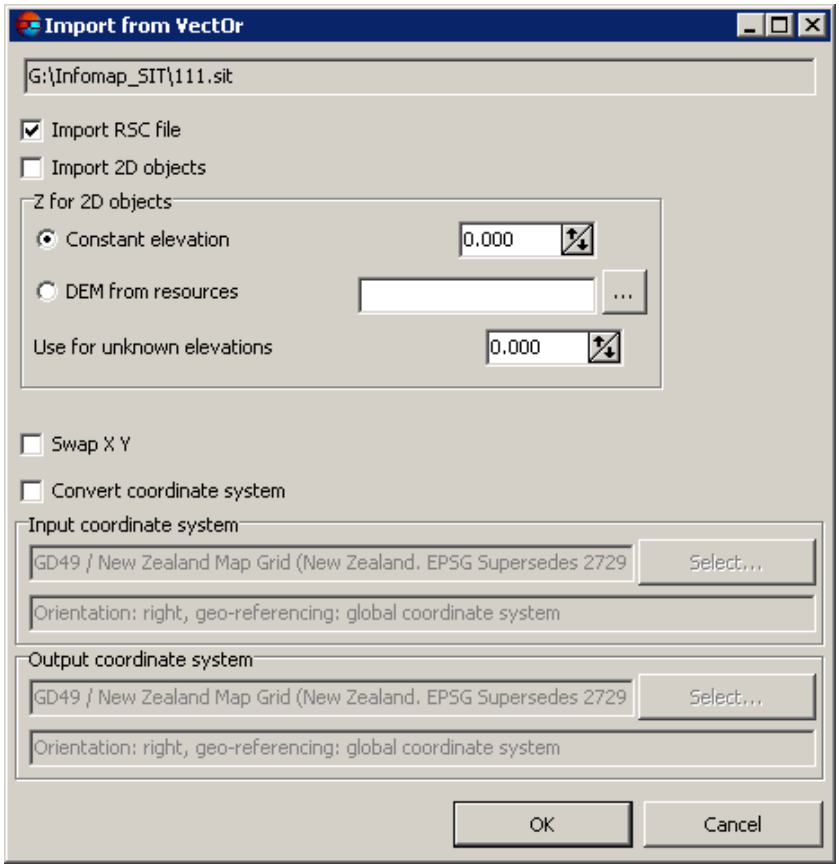
To import 2D shape file, check the option "**Import 2D objects**".
The way to obtain Z coordinates for object points may be chosen from the following options (in "**Z for 2D objects**" pane):

- **Constant elevation** – all objects are assigned the same Z value
- **DEM from resources** – allows to select a resource with DEM to peek Z values from
- **Use for unknown elevations** field defines the Z value to be used for points falling into DEM void cells (cells with no elevation assigned)

Each Shape file has accompanying DBF file, which is used for linking objects to classifier, see section Importing attributes.
The "**Base attributes**" panel is used to define which fields in DBF correspond to basic attributes of the objects: **object name**, **code**, **code name** and **layer name**.

9.1.14. Import from VectOr format

To import data from PHOTOMOD VecrOtr format execute menu command **Vectors | Import | VectOr.** Files with extensions .sit or .map may be imported.
The import parameters dialog for this format looks as follows:



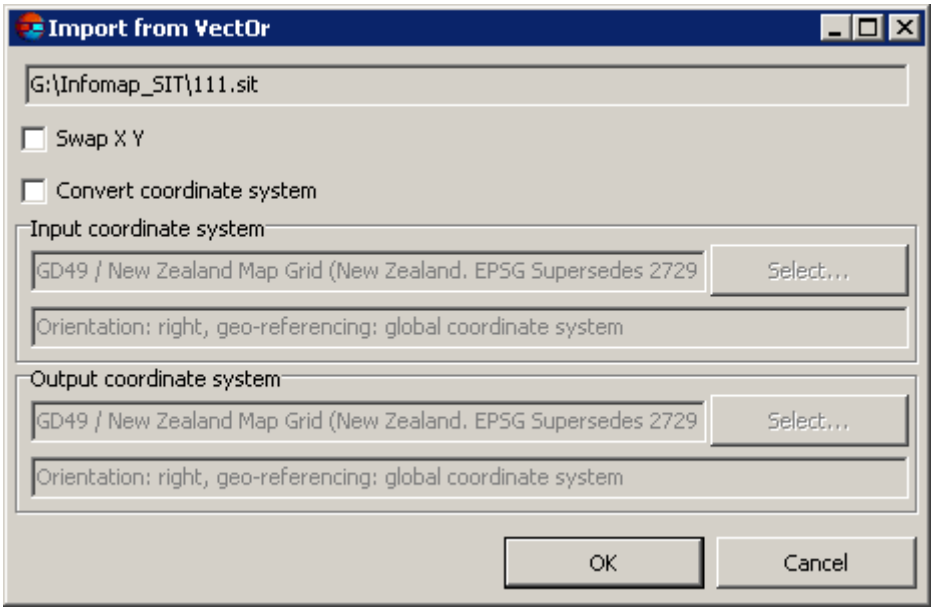
The "**Import RSC file**" option allows converting VectOr classifier used by the imported map to PHOTOMOD classifier, preserving links between objects and code entries.
The "**Import 2D objects**" option defines whether 2D objects are imported and converted to 3D. The way to obtain Z coordinates for object points may be chosen from the following options (in "**Z for 2D objects**" pane):

- **Constant elevation** – all objects are assigned the same Z value
- **DEM from resources** – allows to select a resource with DEM to peek Z values from
- **Use for unknown elevations** field defines the Z value to be used for points falling into DEM void cells (cells with no elevation assigned)

Note: VectOr maps support storing code, code name, object type, layer name, as well as attributes for each object. But the semantic structures of Vector and PHOTOMOD are not fully equivalent, thus export/import operations with VectOr data may lead to loss of certain data.

9.1.15. Import pickets from VectOr format

If data in a VectOr map is needed for DTM creation only, in order to speed up import and decrease the amount of needed RAM, it is advisable to import point objects and line vertices only, without semantic data. This is achieved by using menu command **Vectors | Import | Vector (pickets only)**. Imported pickets are not linked to classifier.
The import parameters dialog has no format-specific options:

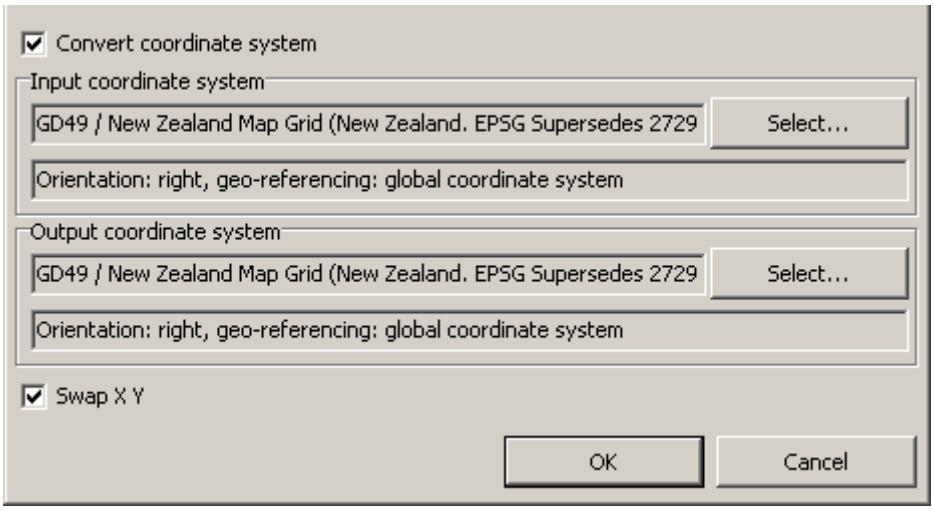


9.2. *Export*

Exporting any vector format begins with the system standard save file dialog for entering the file name to export data to. Then the export parameters dialog is shown for settings parameters of exported coordinates conversion, and parameters specific to the format being exported.
Regardless the format, the parameters dialog has in its upper part a checkbox "**Export selected only**". This checkbox is enabled if the exported layer has selected objects in it. If it is checked, only these selected objects are written to the output file.

9.2.1. Coordinates conversion during export

In the process of export for all formats there are provided options to swap X and Y plane coordinates and to convert vector data from one coordinate system to another. For setting up this conversion, parameters dialog for each format contains in its bottom part a panel as shown on the illustration:

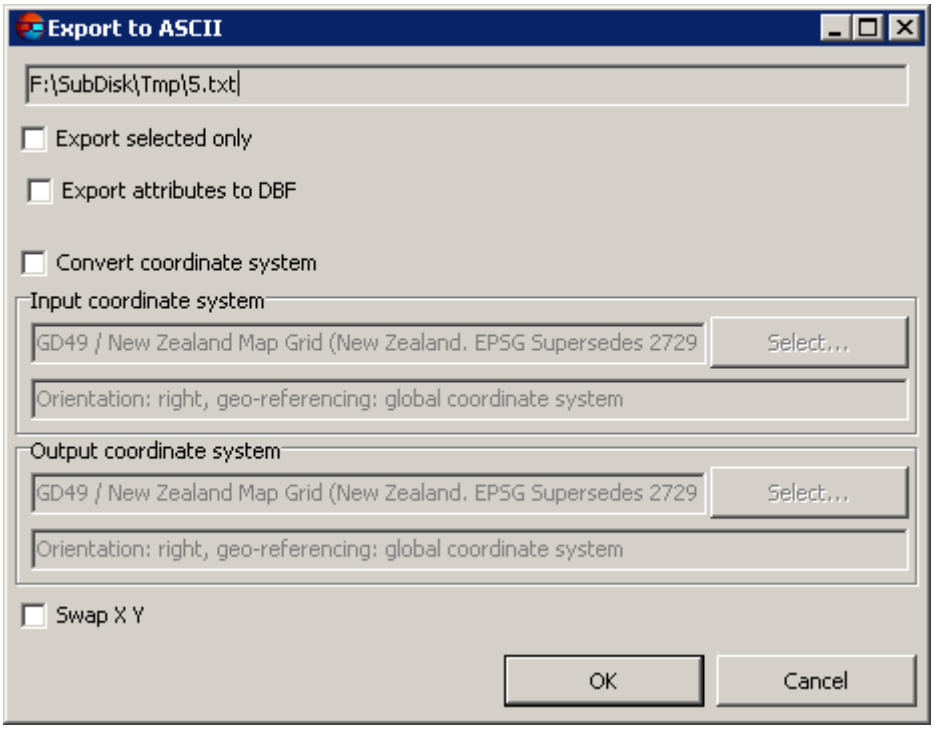

Convert coordinate system
Input coordinate system
GD49 / New Zealand Map Grid (New Zealand. EPSG Supersedes 2729
Select...
Orientation: right, geo-referencing: global coordinate system
Output coordinate system
GD49 / New Zealand Map Grid (New Zealand. EPSG Supersedes 2729
Select...
Orientation: right, geo-referencing: global coordinate system
Swap X Y
OK
Cancel

When "**Swap X Y**" option is checked, plane coordinates of output points are swapped.
When "**Convert coordinate system**" option is checked, input and output coordinate systems are assigned in a fashion common to all PHOTOMOD system – the "Select" button provides for selecting a coordinate system from preinstalled database, from file or from resource. The

default setting for input and output coordinate systems is the coordinate system of the loaded project.
Note that if both options ("**Swap X Y**" and "**Convert coordinate system**") are checked, first of all coordinates of points are converted between coordinate systems and then the plane coordinates of result is swapped.

9.2.2. Export to ASCII format

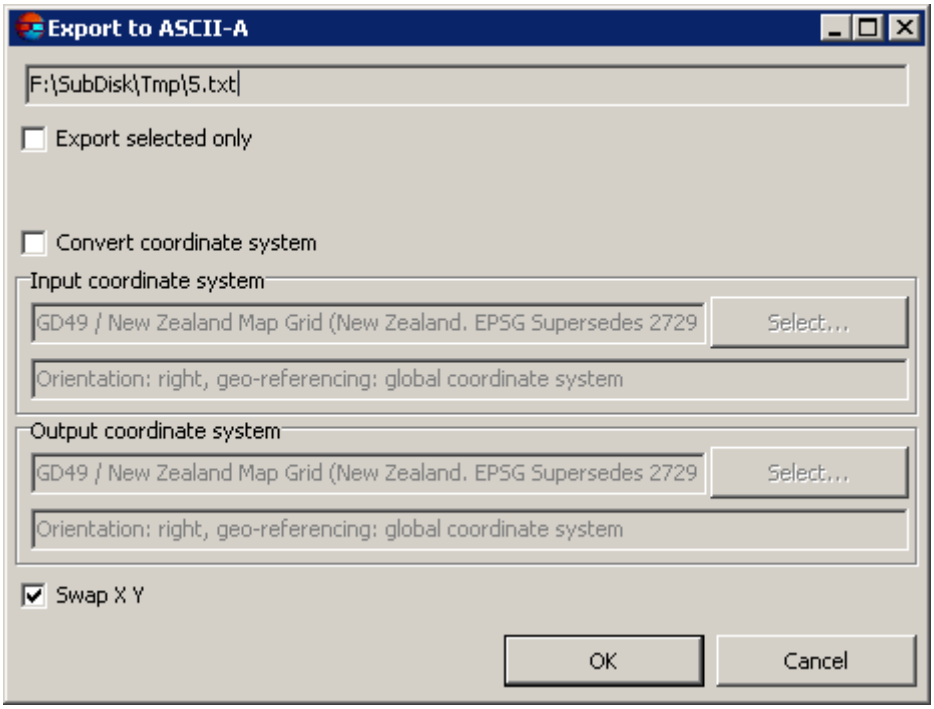
To export data to ASCII format execute menu command **Vectors | Export | ASCII.** Files in this format have default extension .txt. See also format description in section ASCII format description.
The export parameters dialog for this format looks as follows:



The "**Export attributes to DBF**" option allows saving along with the ASCII file containing geometry data, a DBF file (with the same name as ASCII and extension .dbf) containing semantic data. This file stores all attributes for the objects. Besides, for vector layers with classifier, information about code entries linked to objects is also saved to the DBF file.

9.2.3. Export to ASCII-A format

To export data to ASCII-A format execute menu command **Vectors | Export | ASCII-A.** Files in this format have default extension .txt. See also format description in section ASCII-A format description.
The export parameters dialog for this format does not contain any format-specific options:

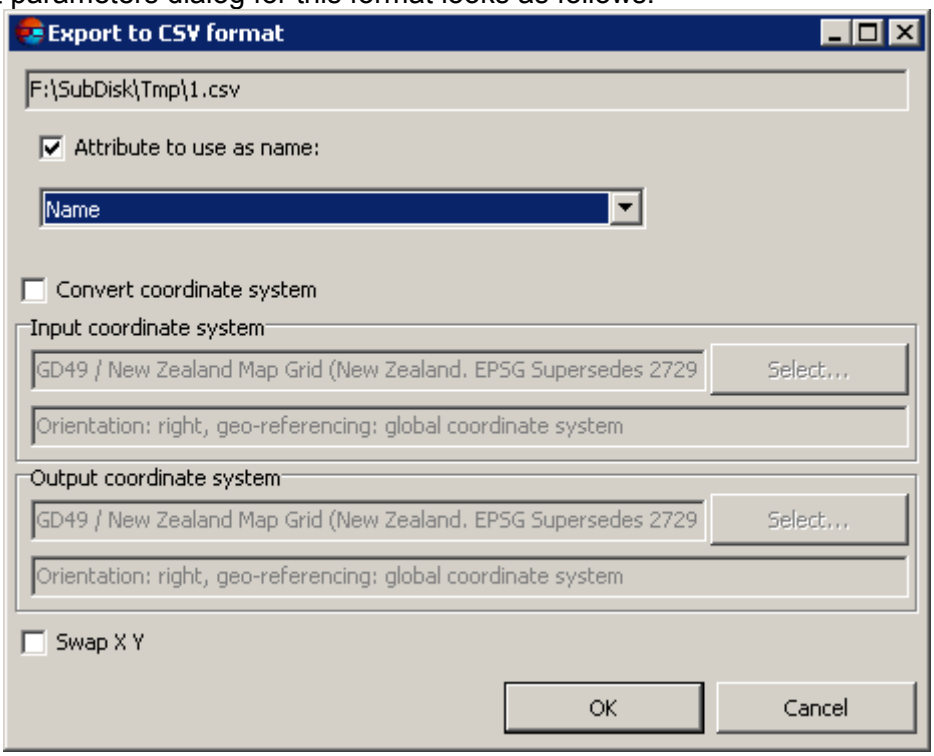


9.2.4. Export to CSV format

To export data to CSV format execute menu command **Vectors | Export | CSV.** Files in this format have default extension .csv.

Export to CSV format in the current PHOTOMOD version is designated for exporting objects actually representing orthophoto sheet frames. Only 4 first vertices (and only X & Y coordinates of them) of each polyline objects are exported along with (optional) the object name.

The export parameters dialog for this format looks as follows:



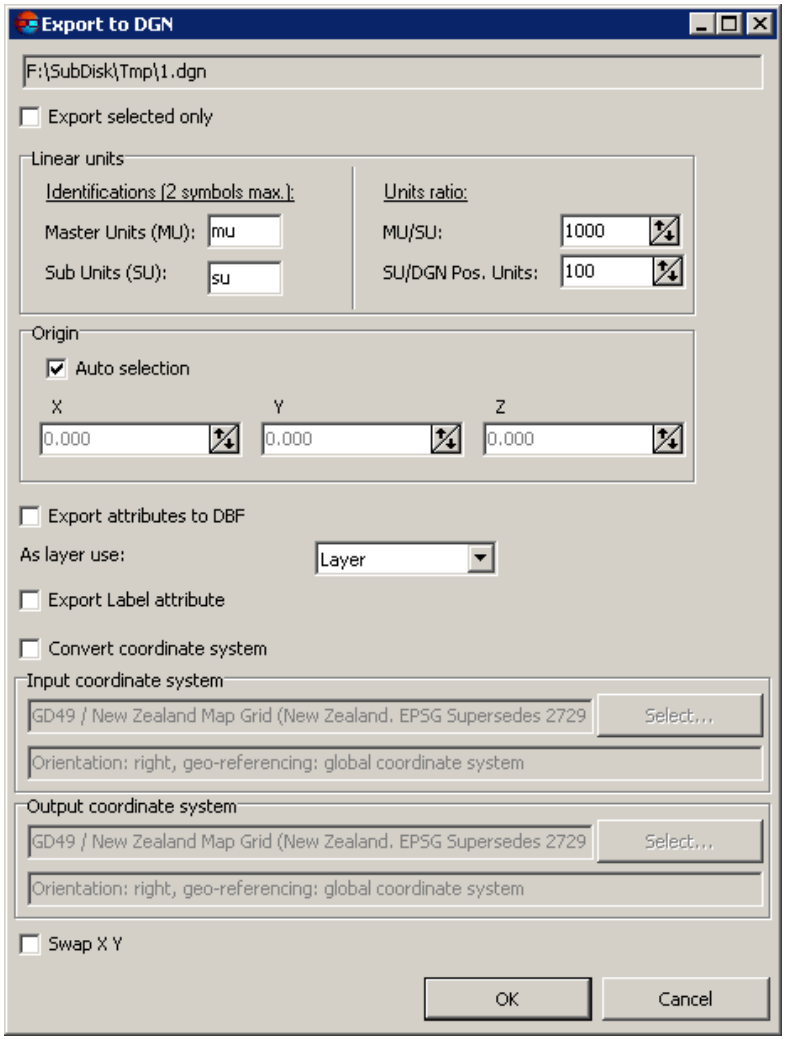
The field "**Attribute to use as name**" contains dropdown list of all existing attributes in a layer the one to be used as an object name in the first field of each line of CSV file. If an

object does not have the attribute with the chosen name, an empty field is saved (i.e. the line would start with the field separator – a comma).
Corresponding import templates (see chapter Import from CSV format) are:

- For checked option " Attribute to use as name ":
  Name, X1, Y1, X2, Y2, X3, Y3, X3, Y4
- For unchecked option " Attribute to use as name ":
  X1, Y1, X2, Y2, X3, Y3, X3, Y4

9.2.5. Export to DGN format

PHOTOMOD support export to DGN V7 format (also known as ISFF DGN or Intergraph DGN), the default format for MicroStation J / 7. **DGN V8 format, native for MicroStation 8 and newer, is not supported.** Exported form PHOTOMOD DGN V7 file can be opened in any MicroStation version, but the format specifics places some limitations on the completeness of data exchange (see section DGN format peculiarities below).
To export data to DGN format execute menu command **Vectors | Export | DGN.**
The export parameters dialog for this format looks as follows:



Settings grouped in the "**Linear units**" panel (Master units, Sub Units, MU / SU, SU / DGN Pos. Units), are described in section DGN format peculiarities below.
The "**Origin**" allows either manual setting of DGN coordinate system origin or automatic calculation ("**Auto selection**" option).

The "**Export attributes to DBF**" option allows saving DBF file (with the same name as DGN file and extension .dbf) with all objects attributes along with the main DGN file, see also section Importing attributes.
Line styles are also saved for line objects and used for data display when the DGN file is opened in MicroStation.
The "**As layer use**" dropdown list allows choosing an option of linking vector objects to layers in the DGN file: by layer name, by object code or by object code name. A supplementary file with .lvl extension is saved along with the DGN file, containing the layer names. When opening DGN in MicroStation, this level file may be also imported using MicroStation command Settings | Level | Names. Then the layer names may be saved into DGN by the File | Save Settings menu command.
The "**Export Label attribute**" option allows exporting objects labels as well.

9.2.5.1. *DGN format peculiarities*

DGN stores for each object its code, code name and type in a reserved file block 512 bytes in size. If there is enough free space left, object attributes are written to this block as well.
Standard coordinate representation for DGN file (without using undisclosed format extensions available to MicroStation application only) contains 32-bit integer values, i.e. all objects coordinates in file are constrained to the range 0 to $4*10^9$ (or $-2*10^9$ to $2*10^9$), moreover without fractional part. For representing fractional numbers, there are introduced secondary units: besides the primary units (position units in DGN terms), sub units (SU) and master units (MU), and

$$1\ SU = N\ Pos.units$$
$$1\ MU = M\ SU,$$

where N and M are arbitrary integer numbers stored as constants in DGN file. MU and SU (but not Pos. units) may be assigned any two-character names (fields "**Master Units (MU)**" и "**Sub Units (SU)**" in dialog). Thus there is a capability of representing coordinates as fixed-point fractional numbers (export procedure in PHOTOMOD is hard-coded to use MU for coordinates). The range of this coordinates is respectively diminished, i.e. each DGN file is a compromise between range and precision of coordinates. The export parameters dialog allows explicit setting of both multipliers: "**SU/DGN Pos. Units**" (N factor in the given formula) and "**MU/SU**" (M factor). For the dialog sample shown in the screenshot,

$$1\ MU=(M*N)=10^5\ Pos.\ units,$$

which gives precision of 10^{-5}, but the range for changes in MU is constrained to approximately $-2*10^4$ to $2*10^4$, which is usually not sufficient for real-world data. That is why the specification of "**Origin**" is also introduced. It is a set of 3 coordinates (X,Y,Z), this time in floating-point format, i.e. without significant limitations, which is added to all coordinates in file expressed in MU. If (X0,Y0,Z0) standing for the origin, resultant coordinates stored in one DGN file may take any values in range

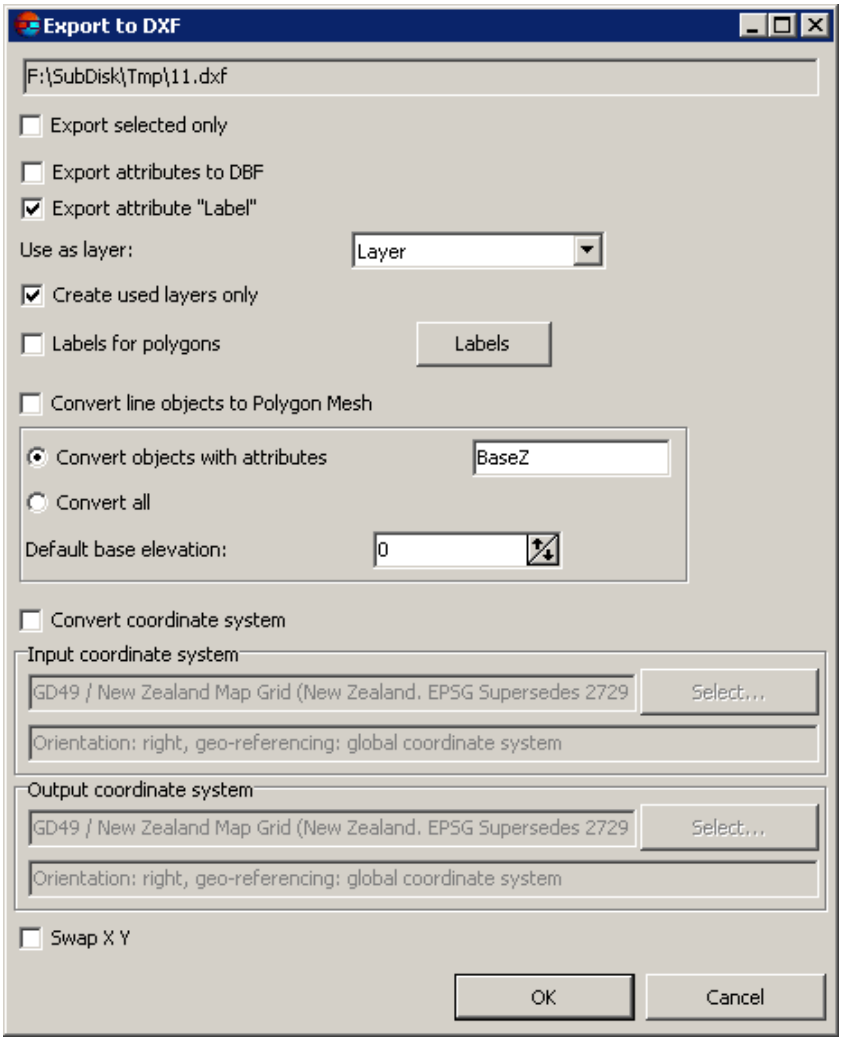
$$\text{from } (X0 - 2*10^9 / M / N\ ;\ Y0 - 2*10^9 / M / N\ ;\ Z0 - 2*10^9 / M / N)$$
$$\text{to } (X0 + 2*10^9 / M / N\ ;\ Y0 + 2*10^9 / M / N\ ;\ Z0 + 2*10^9 / M / N).$$

In PHOTOMOD, this origin may be specified manually, or its calculation may be left for the system (checking the "**Auto selection**" option), which defines it as a center of 3D box circumscribed about all the objects.

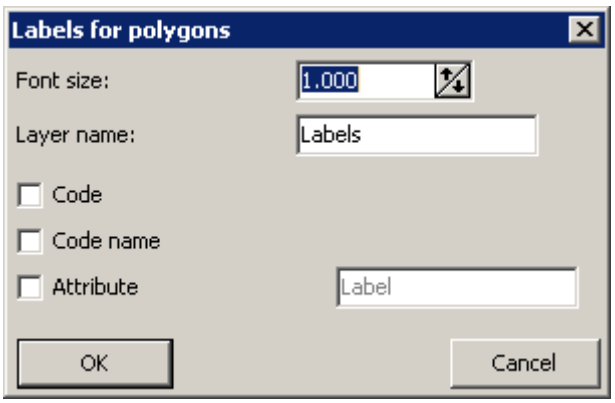
Considering export of classifier layers, each layer is assigned a unique integer index greater then 0. Objects saved to DGN file are assigned a DGN layer with the same index. Layer names are not used inside DGN.

9.2.6. Export to DXF format

To export data to DXF format (exchange format of AutoCAD system) execute menu command **Vectors | Export | DXF.**
DXF version R14 (of AutoCAD R14) is saved.
The export parameters dialog for this format looks as follows:



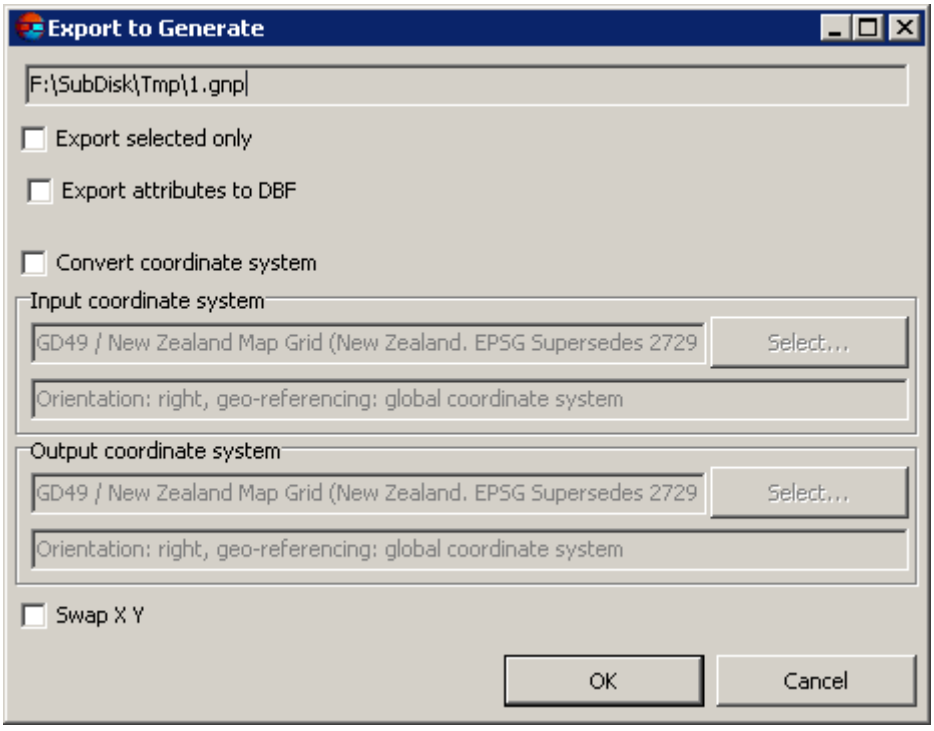
The "**Export attributes to DBF**" option allows saving along with the DXF file containing geometry data, a DBF file (with the same name as DXF and extension .dbf) containing semantic data. This file stores all attributes for the objects. Besides, for vector layers with classifier, information about code entries linked to objects is also saved to the DBF file. See also section Importing attributes.
The "**Export Label attribute**" option allows exporting objects labels as well.
The "**Use as layer**" dropdown list allows choosing the option of assigning objects to layers in the DXF file: by layer name, by object code or by object code name.
The "**Create used layers only**" option defines whether all layers listed in classifier are written, or only those having objects assigned to them.
The "**Labels for polygons**" option allows exporting labels to polygons as a separate layer. Thereat, pressing the "**Labels**" button opens the dialog designated for setting up creation of labels layers. The layer may contain object name, code, code name or attribute. See also section *Creating labels*.



The "**Convert line objects to Polygon Mesh**" option allows converting all line objects (if "**Convert all**" is checked) or only line objects with a certain attribute ("**Convert objects with attributes**") into AutoCAD 3D objects of Polygon Mesh type. Each of such objects is created from the corresponding source line object and its orthogonal projection on a horizontal plane defined by the "**Default base elevation**" field. This feature may be used, for example, for obtaining 3D models of fences having vectorized their upper edges only.
Line styles are preserved during export, and are used for displaying exported objects in AutoCAD.

9.2.7. Export to Generate format

To export data to ArcINFO Generate format execute menu command **Vectors | Export | Generate.**
The export parameters dialog for this format looks as follows:



The "**Export attributes to DBF**" option allows saving along with the file containing geometry data, a DBF file (with the same name as .gnp/.gnl and extension .dbf) containing semantic data. This file stores all attributes for the objects. Besides, for vector layers with classifier,

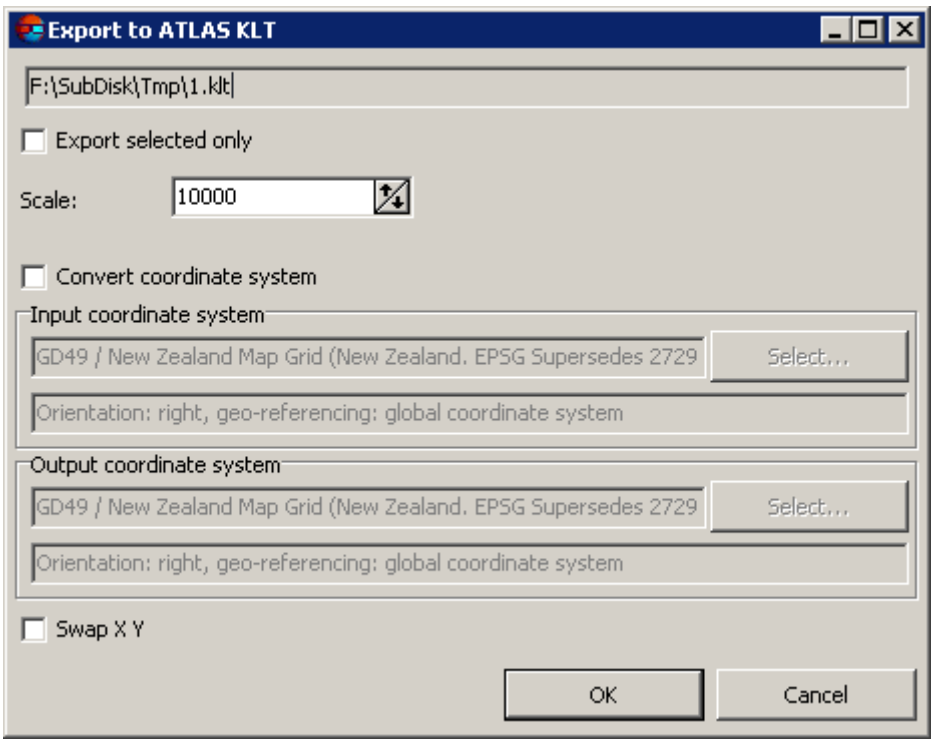
information about code entries linked to objects is also saved to the DBF file. See also section Importing attributes.

If the source layer contains polylines, they are saved to file with .gnl extensions. Point are exported to a file with the same name and .gnp extension.

During export to Arc Generate each object is assigned an integer index. If object code in PHOTOMOD is an integer, it is used as this index. Otherwise the object is assigned an index greater by 1 then the maximum of all used indices.

9.2.8. Export to ATLAS KLT format

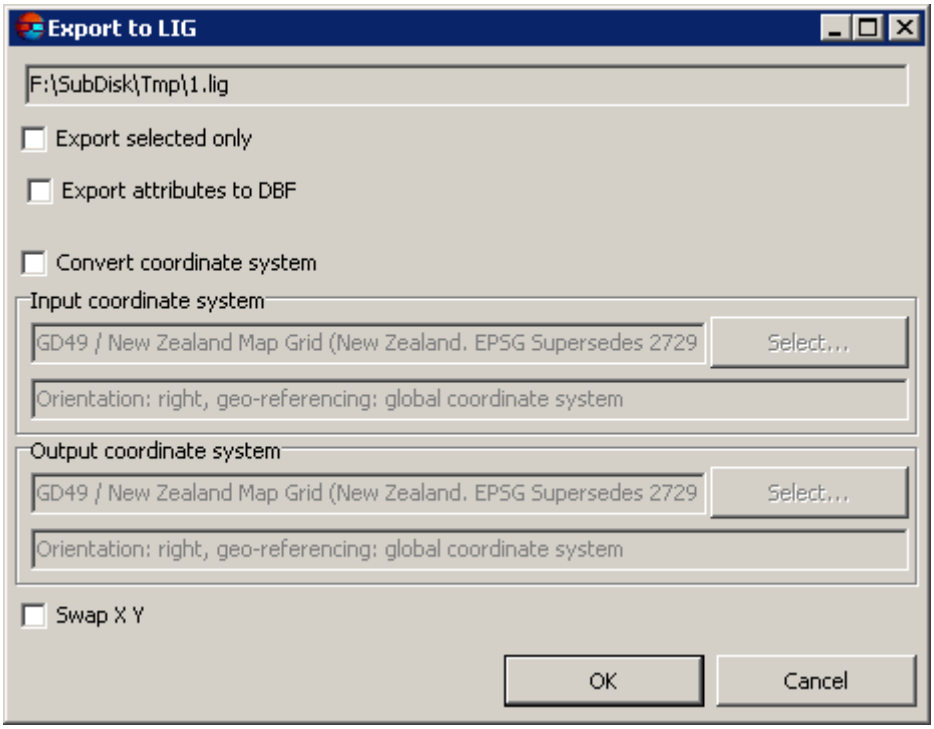
To export data to ATLAS KLT format execute menu command **Vectors | Export | ATLAS KLT.**
The export parameters dialog for this format looks as follows:



The "**Scale**" field defines the scale text that is written to the header of KLT file and has no other impact on the export process.

9.2.9. Export to LIG format

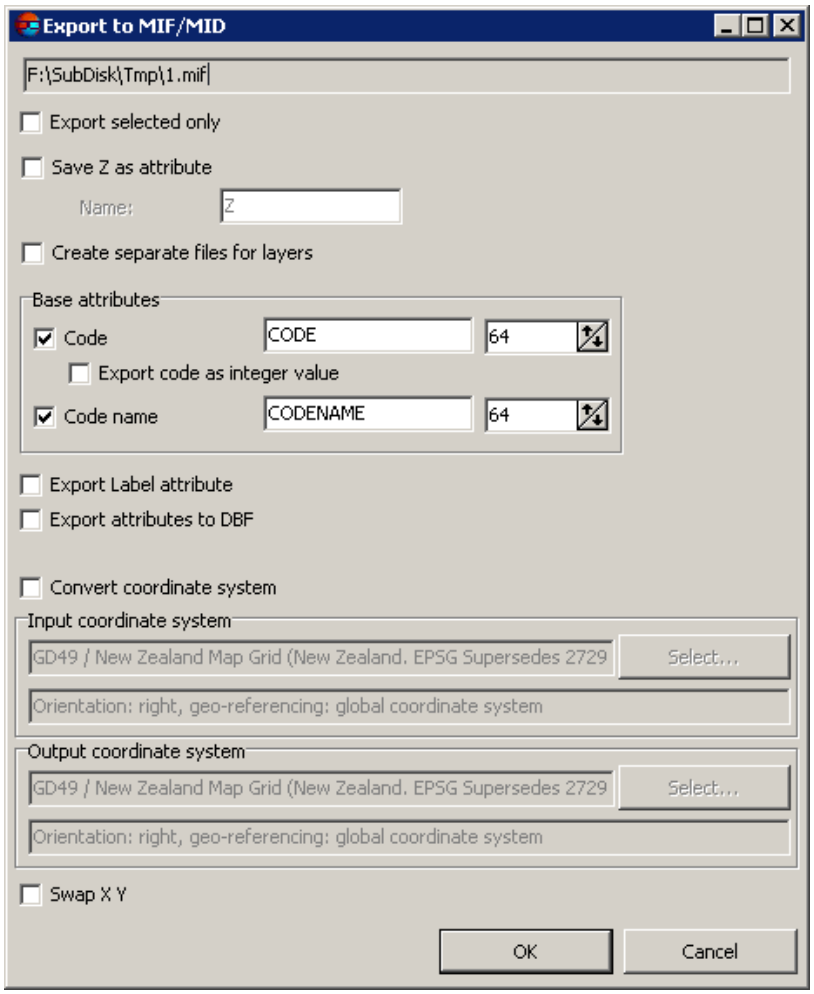
To export data to LIG format execute menu command **Vectors | Export | LIG.**
The export parameters dialog for this format looks as follows:



The "**Export attributes to DBF**" option allows saving along with the file containing geometry data, a DBF file (with the same name as .lig and extension .dbf) containing semantic data. This file stores all attributes for the objects. Besides, for vector layers with classifier, information about code entries linked to objects is also saved to the DBF file. See also section Importing attributes.

9.2.10. Export to MIF/MID format

To export data to MIF/MID format (MapInfo exchange format) execute menu command **Vectors | Export | MIF/MID.**
The export parameters dialog for this format looks as follows:



The "**Save Z as attribute**" option allows saving Z coordinate as an attribute in the .mid file. The name of the attribute is defined in the adjacent field "**Name**" (default – "Z").
The "Create separate files for layers" allows exporting objects from each classifier layer into separate file with the name of this layer. If this option is not checked, all objects will be exported to one file.

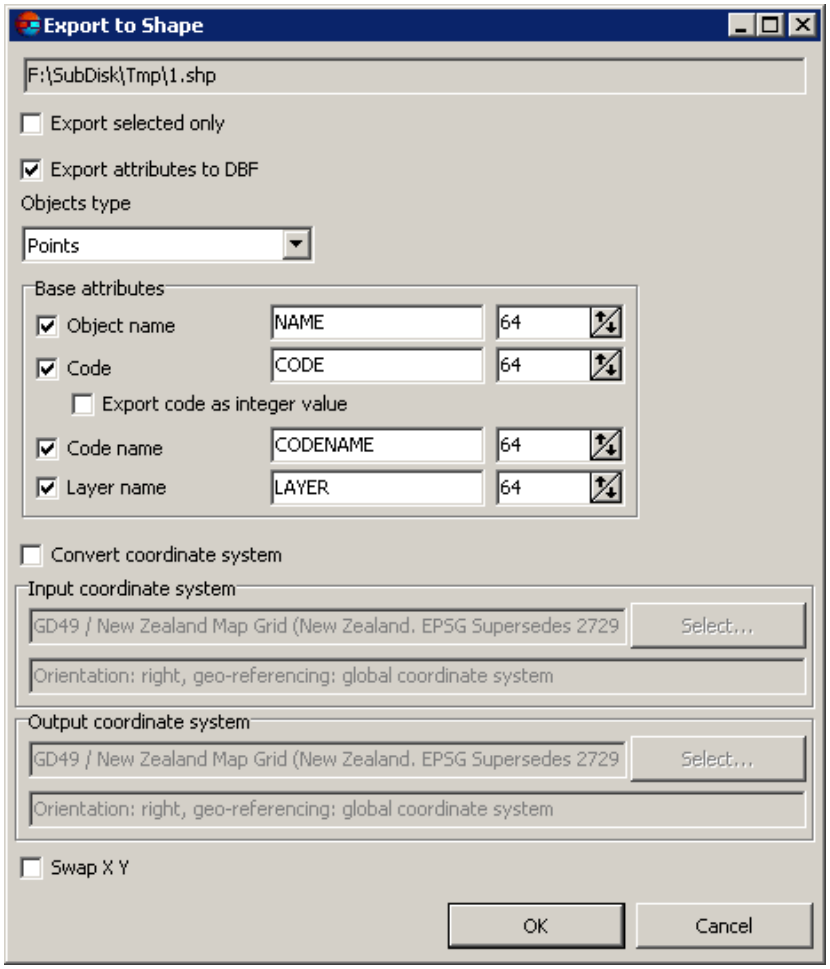
PHOTOMOD may use a text line of arbitrary length as an attribute. MIF/MID files may contain fixed length lines only. So the **Base attributes** panel is used to set sizes for basic fields **Code** and **Code name** (default value is 64).

If there are PHOTOMOD objects with codes not being integers numbers, it is necessary to turn the option "**Export code as integer value**" on, otherwise these objects may be not shown in MapInfo.
The "**Export Label attribute**" option allows exporting objects labels as well.
The "**Export attributes to DBF**" option allows saving along with the file containing geometry data, a DBF file (with the same name as .mif/.mid and extension .dbf) containing semantic data. This file stores all attributes for the objects. Besides, for vector layers with classifier, information about code entries linked to objects is also saved to the DBF file. See also section Importing attributes.

Two file are created as the result of the export: with identical names and extensions .mif storing geometry of vector objects, and .mid with attributes.
MIF/MID format does not allow spaces in column (attribute) names, so during export in PHOTOMOD all spaces in attribute names are replaced with underscores, while during import – underscores are replaced with spaces.

9.2.11. Export to Shape format

To export data to ArcINFO Shape format execute menu command **Vectors | Export | Shape.**
The export parameters dialog for this format looks as follows:



The "**Objects type**" dropdown list: Shape format allows storing of different types of objects in separate files, and the export must be performed for different object types separately, selecting the corresponding type (points, polylines, polygons).

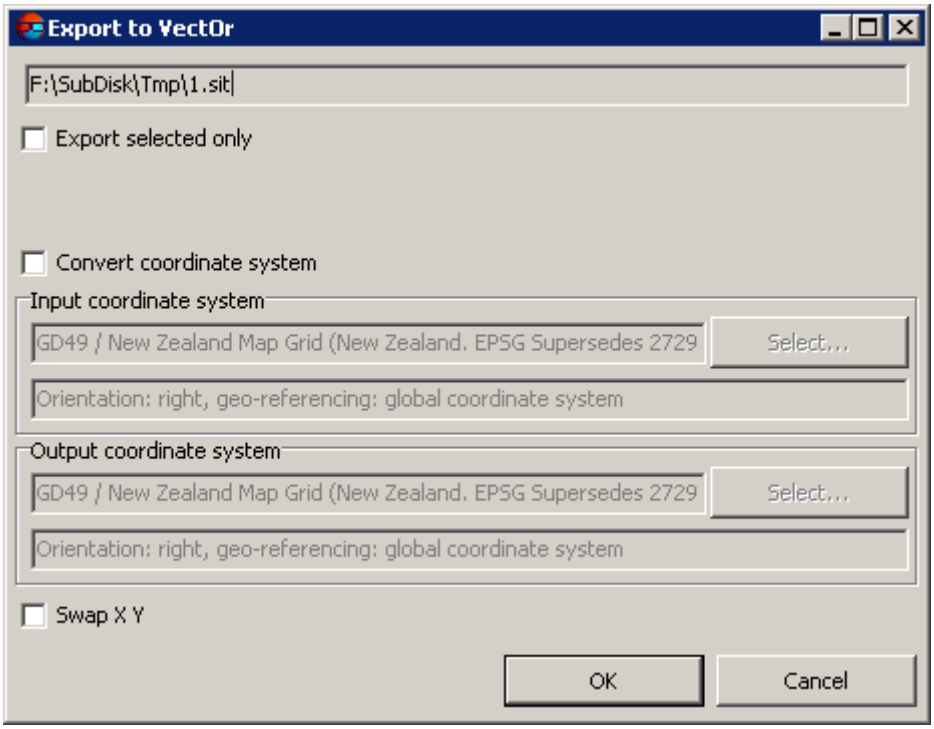
Format specifics:
Each set of Shape files contains at least three files with the same name and different extensions.
*.shp – main file containing objects geometry. One .shp file contains only either points or lines, or polygons.
*.shx – an optional index file containing information on placement of objects in the main file. Is used to speed up reading of the main file.
*.dbf – file containing DBF table. See section Importing attributes.

The "**Base attributes**" panel is used to assign DBF field names for basic PHOTOMOD attributes: object name, code, code name and layer name.

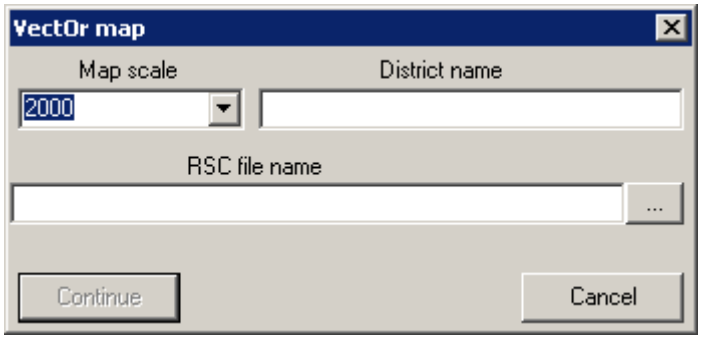
9.2.12. Export to VectOr format

To export data to PHOTOMOD VectOr format execute menu command **Vectors | Export | VectOr.**

The export parameters dialog for this format looks as follows:



If a new SIT file (new map) is being created, after the "OK" button is pressed in the main parameters dialog, an additional dialog is opened to enter map scale, district name (arbitrary text) and RSC file name (classifier name) – a file with *.rsc extension.



Several common classifiers are included in PHOTOMOD distribution and are located in the VectOr subfolder of PHOTOMOD main program folder. See also *PHOTOMOD VectOr User Manual*. If codes in VectOr and PHOTOMOD classifiers match, the objects are assigned to the VecrOt classifier codes and will be displayed in VectOr with styles defined in the RSC file. See also *Importing classifier section*.

9.3. *Import of DEM*

It is possible to import the DEM from the raster format files using the **DEM | Import** menu command:

- Surfer grid (*.GRD)
- ArcInfo grid (*.GRD)
- GeoTIFF (*.TIF)
- USGS DEM (*.DEM)
- VectOr MTW (*.MTW)
- DTED (*.DT?, where ? – character)

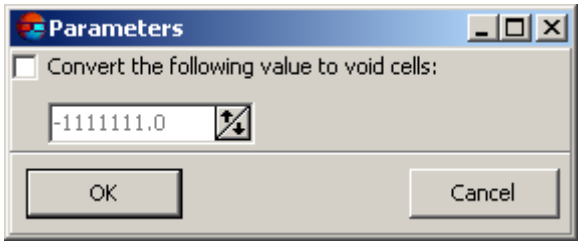
- ERDAS Imagine (*.IMG)
- PCIDSK (*.PIX)
- SRTM (*.HGT)
- GTOPO (*.DEM)

The DEM file selection window of above-listed types opens on command execution. After file selecting and pushing the **Open** button the parameters import window opens, after configuring which, it is suggested to save the DEM in the PHOTOMOD internal format with *.x-dem extension in the resources of active profile. Select the file name and resources folder for saving and press OK. Define the method of loading - into the active DEM layer (if there already is such layer in Layer Manager) or into a new layer.

It is possible to import the DEM as a set of points. To do this, import the file which contains the DEM in the ASCII format, into the *Vectors* layer using the **Vectors | Import | ASCII** main menu command. At that the source file may also be in csv format (set of X, Y, Z coordinates of DEM elements, separated by commas or spaces), but with the *.txt extension.

9.3.1. Batch DEM import

Feature of batch DEM import is provided in the system. It allows in the same session to transform the few files with DEMs of any supported format (see list of formats in the chapter Import of DEM) to DEM internal formats files, which are saved into the resources of active profile.
Use the **DEM | Batch import** menu command to batch import. **Open** window opens to select the files with DEMs. Select the files with DEMs and push the **OK** button. **Parameters** window opens, which allows to set the parameters for output DEMs, namely the cells with given Z value transformation to void cells.



Push the **OK** button. Window of selection the active profile folder to saving the files with output DEMs. The name of output DEMs files is the same as of sources files except for the extension which is replaced with x-dem.

9.4. *Export of DEM*

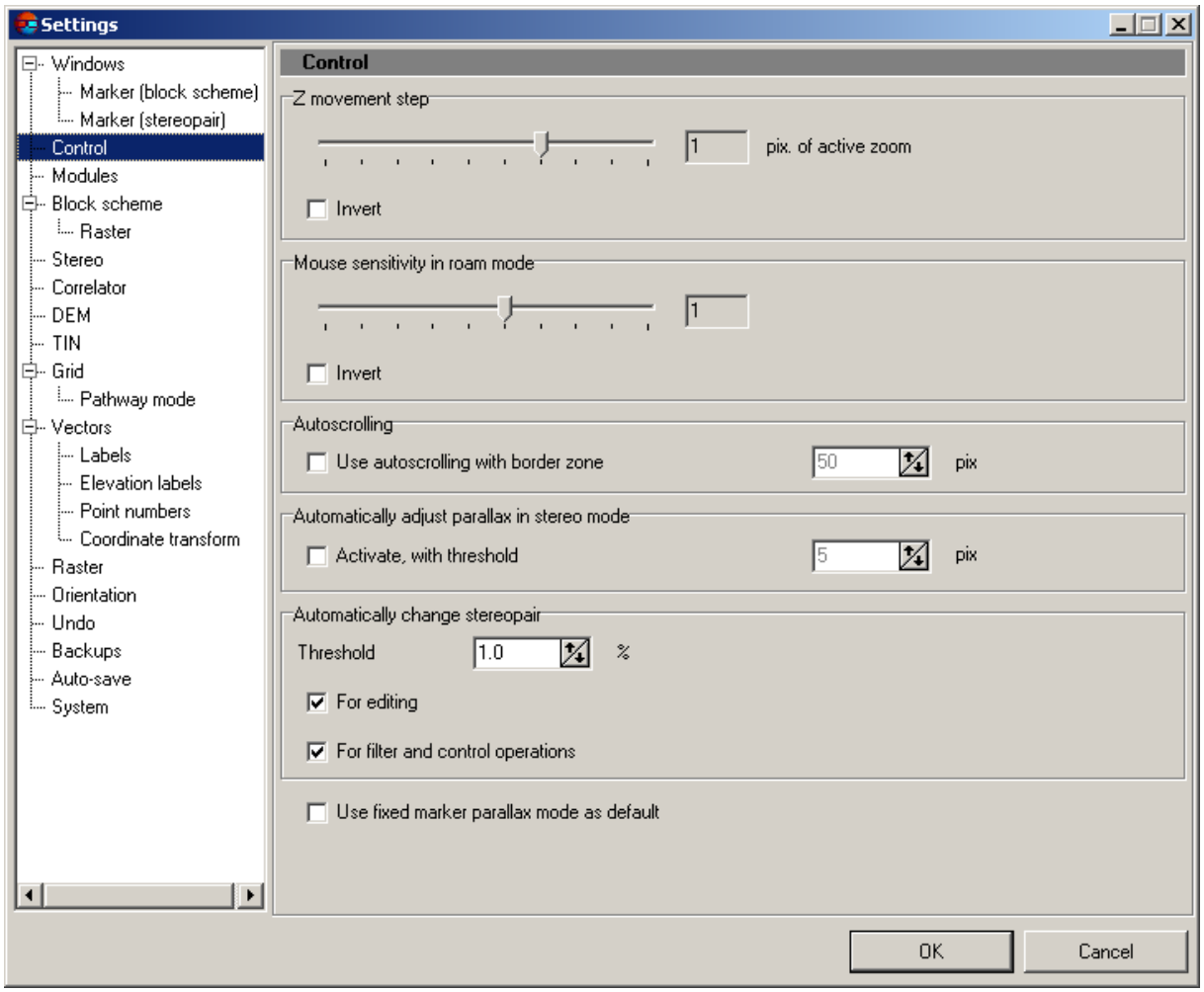
The **PHOTOMOD** system supports a number of formats to export the DEM:

- ArcInfo ASCII grid
- ASCII
- CSV
- DTED DEM
- ERDAS Imagine DEM
- GeoTIFF DEM
- VectOr MTW
- PCIDSK DEM
- Surfer ASCII grid
- USGS DEM

Select the DEM for export from the Layer Manager (make this layer active), and then use **Vectors | Export |** (export format) menu command. The dialog appears for choosing file name (outside the resource profile) to save exported DEM to. Press the **Save** button. The export parameters window opens. Set the parameters and press **OK**.

10. Parameters and settings

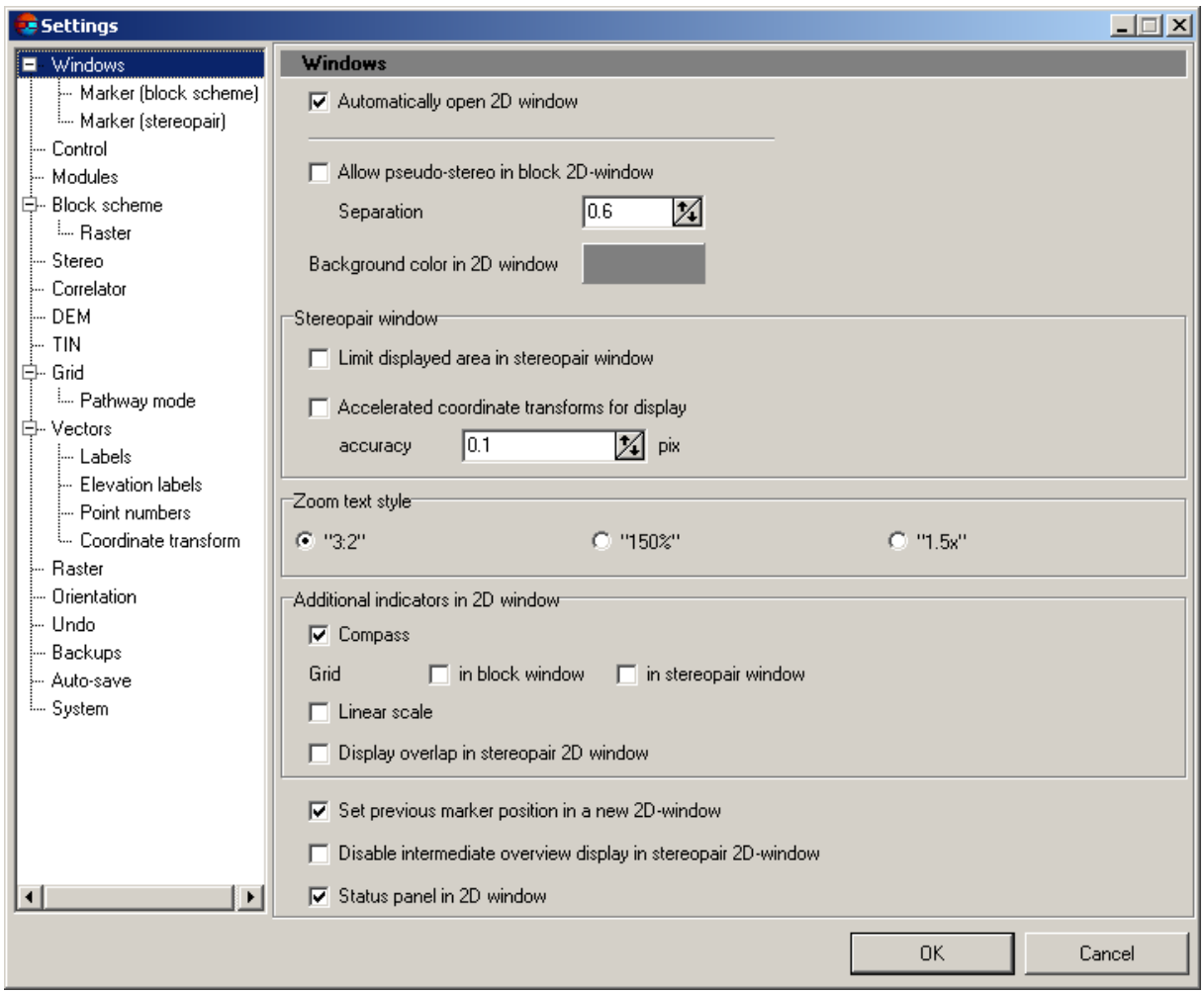
Settings window, which contains tabs set, which allow to adjust the **PHOTOMOD** system parameters for various steps performing of project processing, opens by the **Service | Settings** command (or by pushing the button of **PHOTOMOD Core** main toolbar).



The Settings window

10.1. *Windows settings*

The tab **Windows** in the **Settings** window (menu **Service | Settings** or the button) allows setting 2D-windows parameters.



2D-window parameters

- **Automatically open 2D window** (switched on by default) – provided it is switched off during the next **PHOTOMOD** session only main menu and toolbars used in previous session will be opened;
- **Allow pseudo-stereo in block 2D window** – allows to enable stereo mode in block scheme window. After selecting the option you should close and then re-open the block scheme window – you will be allowed to turn on the stereo mode there, at that for the left eye an orthogonal projection is used, and for the right one – parallel projection with some angle to normal line. The angle is defined by the **Separation** parameter.
- **Background color in 2D window** – allows changing colour of the background of all opened 2D-windows (using mouse double-click)

The section **Stereopair window** contains the following options:

- **Limit displayed area in the stereopair window** – allows displaying only that parts of vector layers that fall into stereopair limiting parallelepiped (with respect to the project reference system) which accelerates the objects displaying.
- **Accelerated coordinates transforms for display with the accuracy** – allows accelerating displaying working area of the stereopair window (the system tries to approximate transforming from the project reference system to pixels using piecewise-projective transformation with accuracy specified in pixels).

In the section **Zoom text style** it is possible to select one of the three available scale displaying styles.

The section **Additional indicators in 2D window** contains the following options:

- **Compass** – allows displaying orientation of the axes of the project reference system at the marker position at the left bottom corner of the 2D-window.
- **Grid** – allows displaying coordinate grid in the 2D-windows of the block and/or stereopair. In the stereowindow the grid will be displayed approximately at the marker height.
- **Linear scale** – allows displaying of the scale (using the marker coordinates) in the left bottom corner of the 2D-window.
- **Display overlap in stereopair 2D window** – allows displaying the images overlapping area in the stereopair 2D-window

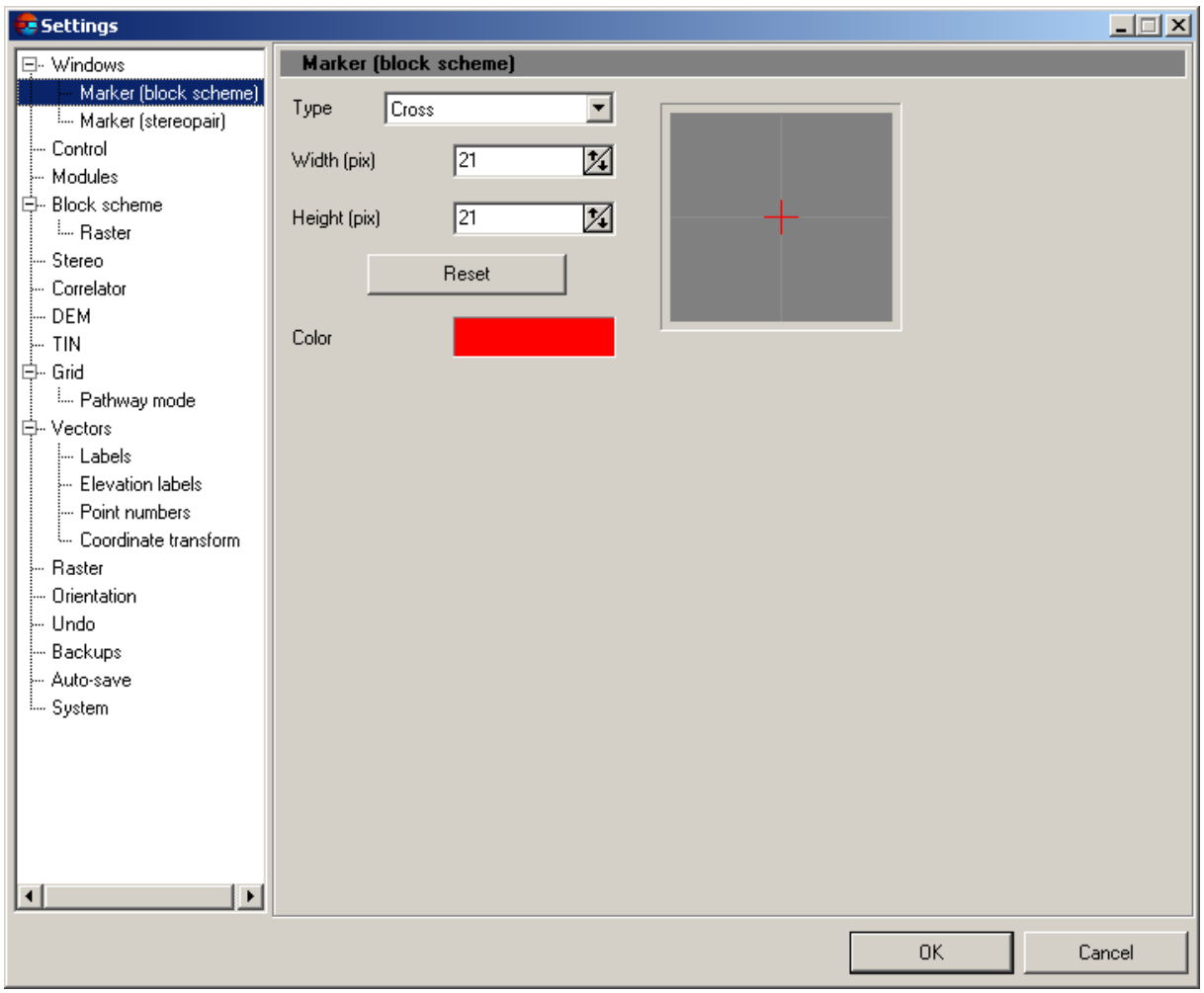
Remark: Compass, Linear scale and Grid are placed in the Marker layer, and they could be hidden (along with the marker) by switching off the layer visibility. See chapter Layer manager.

- The option **Set previous marker position in a new 2D window** – when new 2D-window is opened using menu commands **Windows | New 2D-window (block)** and **Windows | New 2D-window (stereopair)** the marker will be placed at the same position on the image as in the previous window.
- The option **Disable intermediate overview display in stereopair 2D window** – allows accelerating stereopairs changing with possible loss of visual comfort.
- The option **Status panel in 2D window** – allows displaying/hiding status panel at the bottom of 2D-window.

See also the chapter Main windows.

10.2. *Marker settings*

Windows | Marker tab in **Settings** window (**Service | Settings**) or the button allows setting the following parameters of marker displaying on a block scheme or on a stereopair:



Marker displaying parameters

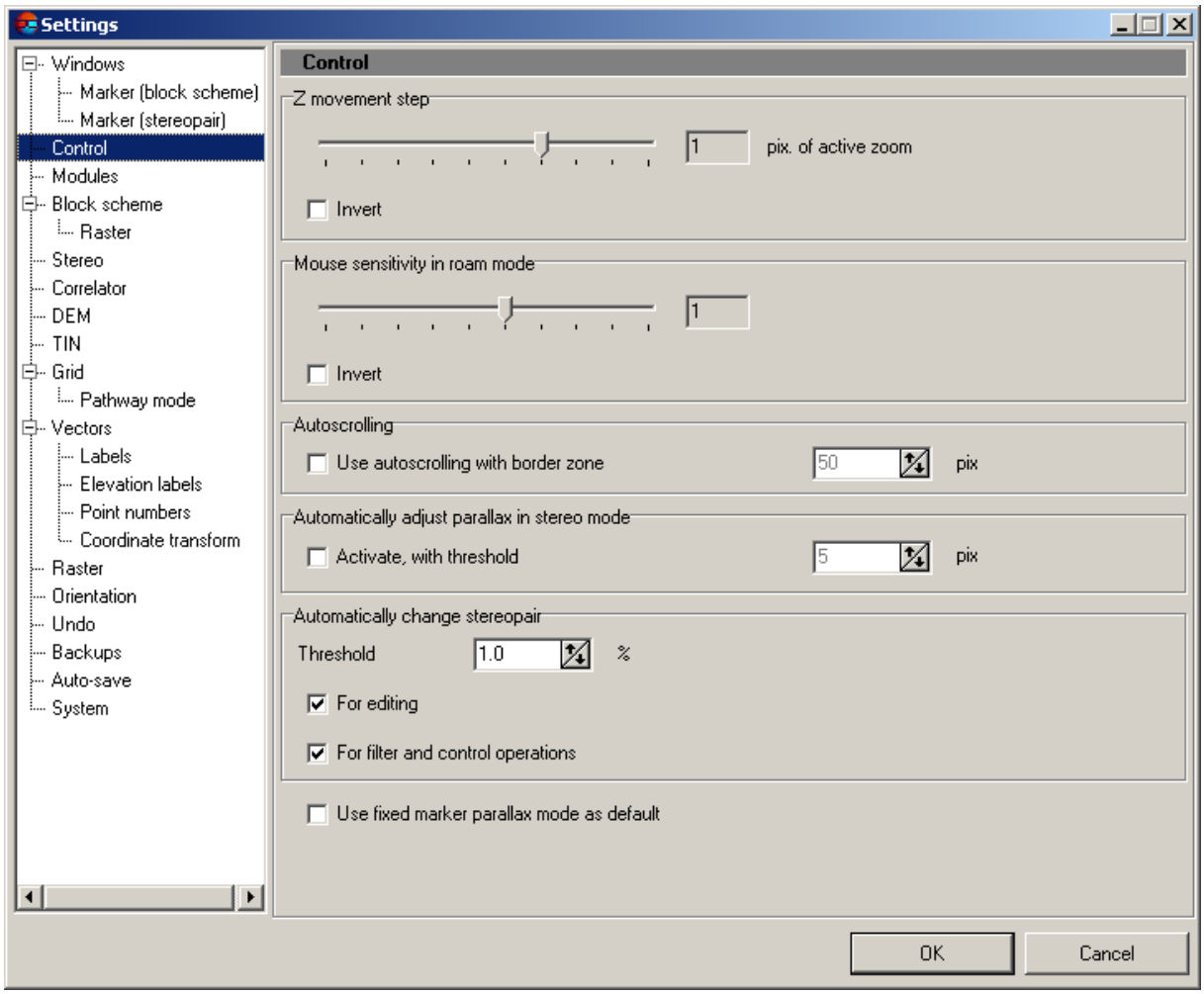
You can change the following parameters of marker displaying:

- **Type** – marker shape (Cross, T (four types), Aim, Rectangle (four types))
- **Width** – marker width in pixels
- **Height** – marker height in pixels
- **Color** – marker color

See also the chapter Operating marker.

10.3. *Control settings*

Control parameters are set up on the tab **Control** in the window **Parameters** (menu **Service | Parameters** or the button):



Control settings

- **Z movement step** – allows to set Z step of mouse movement using scroller (in pixels in the current scale)
- **Mouse sensitivity in roam mode** – allows adjusting work in still marker mode, defining the rate of image shifting in block or stereopair 2D windows in response to mouse movement.

Autoscrolling section – allows marginal zone defining (in pixels), on entering which autoscrolling starts.

Automatically adjust parallax in stereo mode section – allows defining threshold value (in pixel) to perform automatic parallax adjustment in stereo mode.

Section **Automatically change stereopair** – allows adjusting of automatic stereopair change – by defining the marginal zone (distance from overlapping area margin in percents of image size), on entering which the next stereopair will be made active as well as enabling automatic stereopair change for different modes or operations:

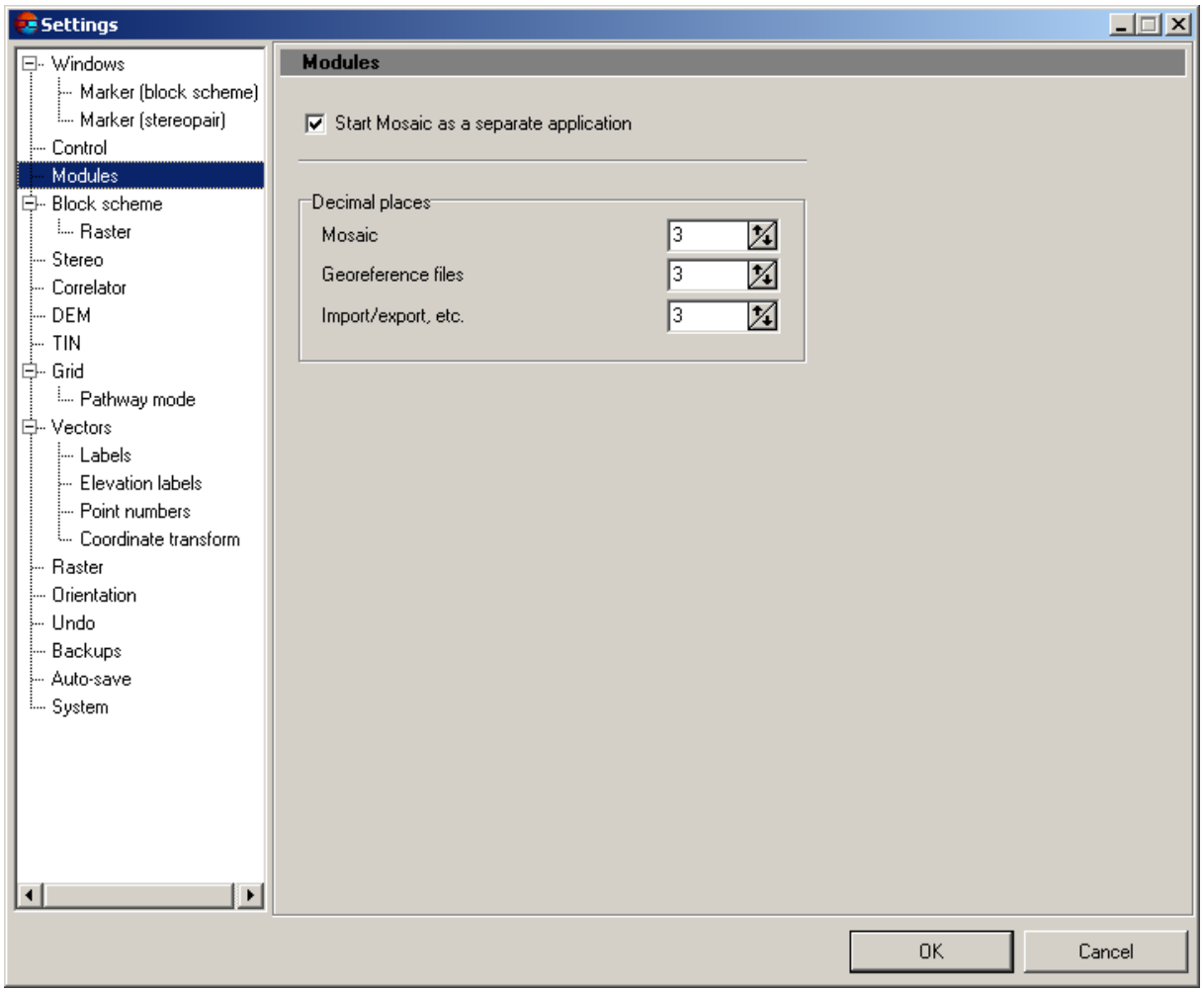
- In the vector object creating/editing mode (**For editing** option);
- In filtering and checking of vector objects, contour lines, TIN, DEMs (**For filter and control operations** options);

Note: Automatic stereopair change in pickets creating in profiling mode (see chapter Creation of pickets in pathway mode) is adjusted on the appropriate tab (see chapter Pathway mode settings).

- **Use fixed marker parallax mode as default** – allows setting the mode by default.

10.4. *Settings of modules starting*

It is possible to set the **Mosaic** module starting in a separate window (**Project | Mosaic** menu command or button) on the **Modules** tab in **Settings** window (**Service | Settings** menu command or button); i.e. **PHOTOMOD Core** window will be available for work, as well as it is possible to define the precision in **Mosaic** module during the georeference, import/export and other operations.



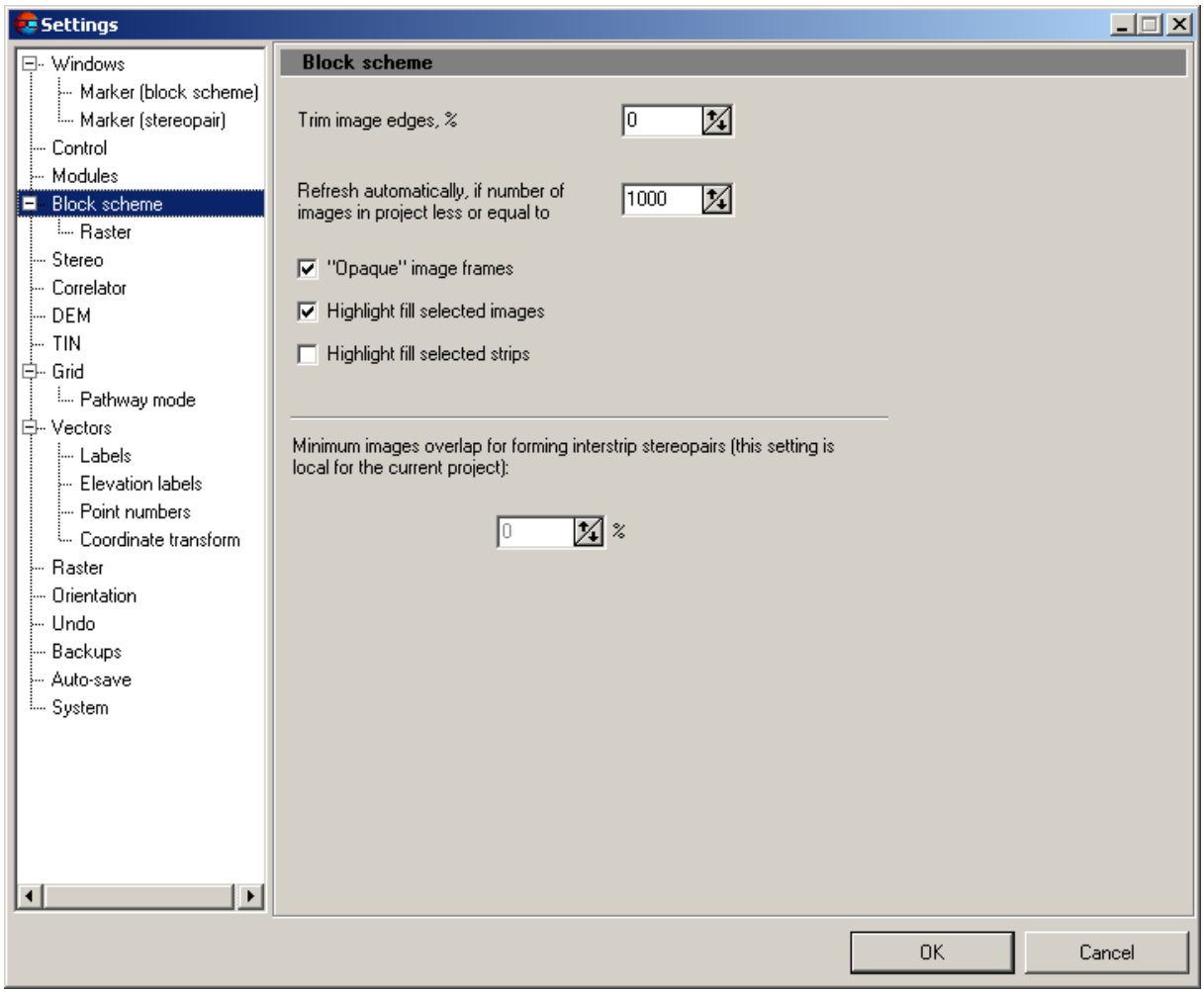
10.5. *Settings of block layout*

The **Block scheme** tab in the **Settings** window (**Service | Settings** menu command or button) allows setting the block scheme displaying and contains the following options:

- **Trim image edges %** - allows to "cut" (hide) the image edges (for example, may be useful in the presence of brownfields in the images of analog cameras);
- **Refresh automatically, if number of images in project less or equal to** - allows to specify the maximum number of images with the which the block scheme will be automatically updated (for example, after editing the tie points or block adjustment);
- **"Opaque" image frame** - option allows setting the displaying only of the visible parts of the framework in accordance with the order of images and strips in the block;
- **Highlight fill selected images** - option allows setting a translucent fill for the selected images;

- **Highlight fill selected stripes** - option allows setting a translucent fill for the selected strips;

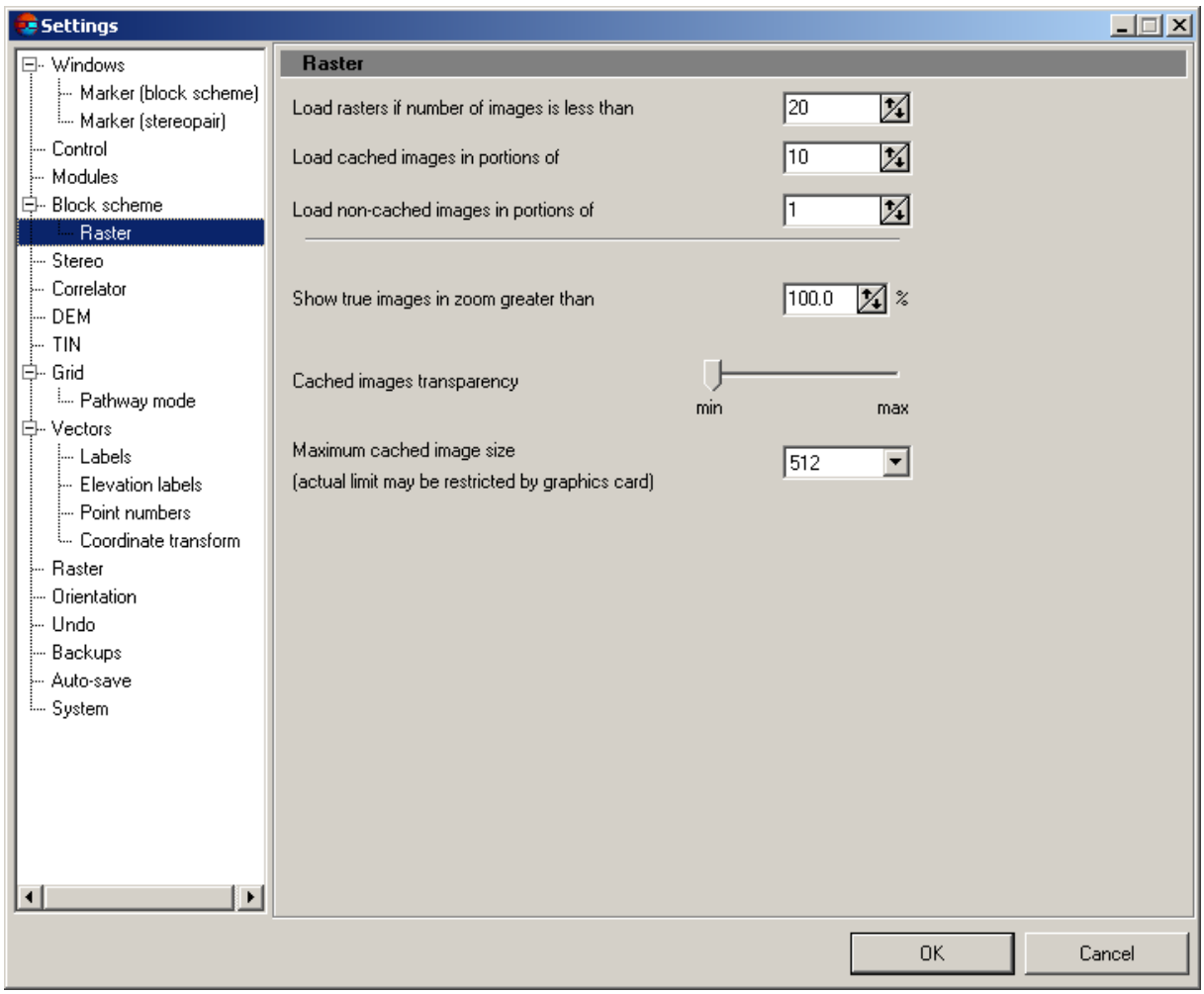
Minimum images overlap for forming interstrip stereopairs (this setting is local for the current project) – it is recommended to set the value not less then 20% for large projects with "good" overlap - 40-50%



Parameters of block scheme displaying

10.5.1. Parameters of loading and displaying images

Raster tab in the **Settings** window (**Service | Settings** menu command or button) allows to set the loading rasters parameters in 2D windows.



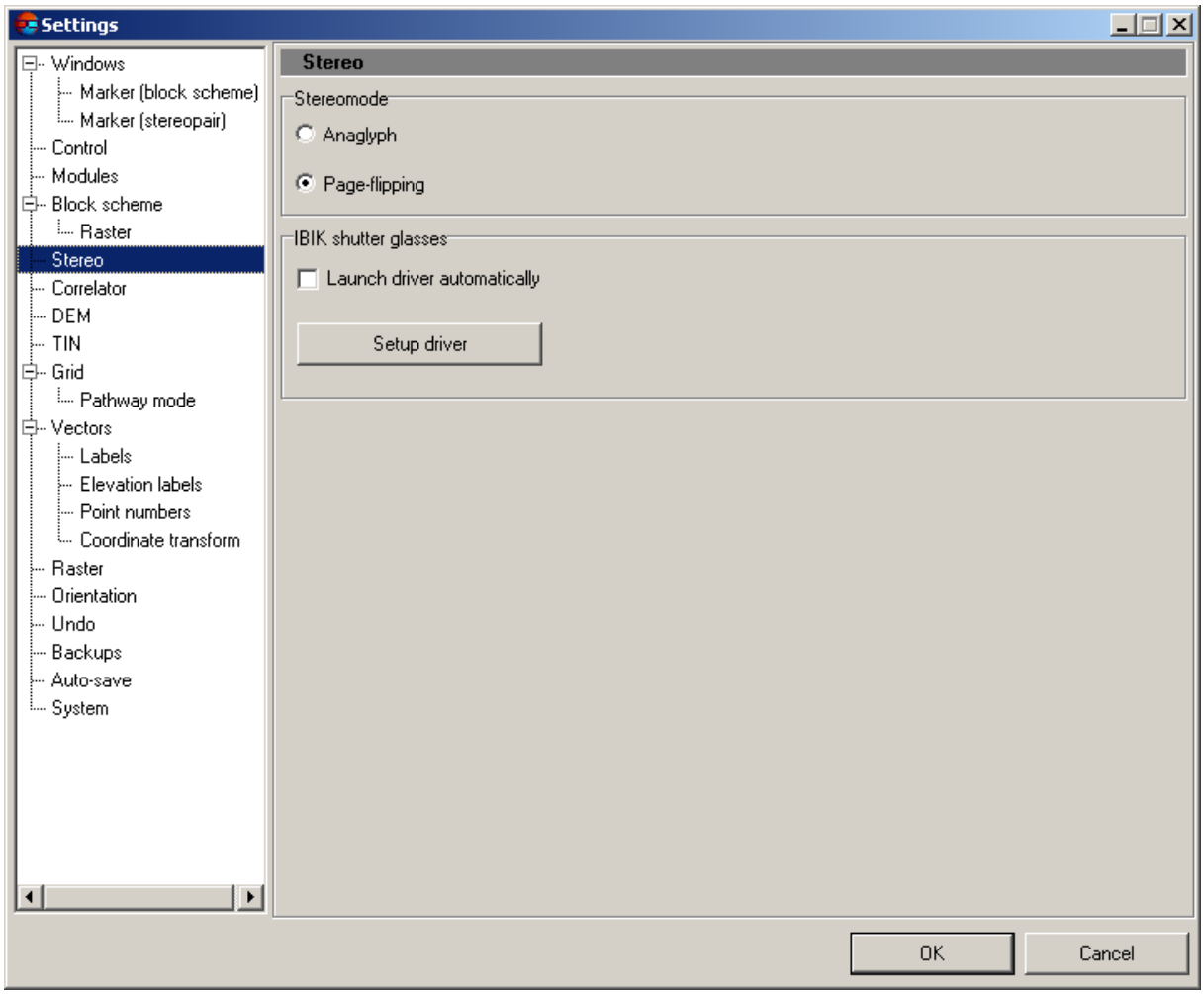
Loading rasters pameters

- **Load rasters if number of images is less then** specified values option – do not display rasters (only frames) if their number in the project is greater then specified in this field. **Block | Show images** menu command allows to display all the rasters regardless of their quantity, though it slows down working with the project.
- **Load cached images in portions of** – allows to set the number of cached images (small copies of images, which are storing in cache) during the rasters loading.
- **Load non-cached images in portions of** – allows to set the number of non-cached images during the loading.
- **Show true images in zoom greater then** – allows to set the threshold zoom, greater which the rasters will be displayed.
- **Cached images transparency** – allows to set the transparency range during the displaying of cached rasters (small copies of images) using the slider.
- **Maximum cached image size** – allows to set the size in pixels for cached raster. The less is the value, the faster are rasters display; the larger is value, the higher the raster display quality.

10.6. *Stereo settings*

Stereo tab in Settings window (Service **| Settings** menu command or button) allows to select the stereomode – Anaglyph or Page-flipping, as well as set and start automatically IBIK shutter glasses driver in case of its usage.

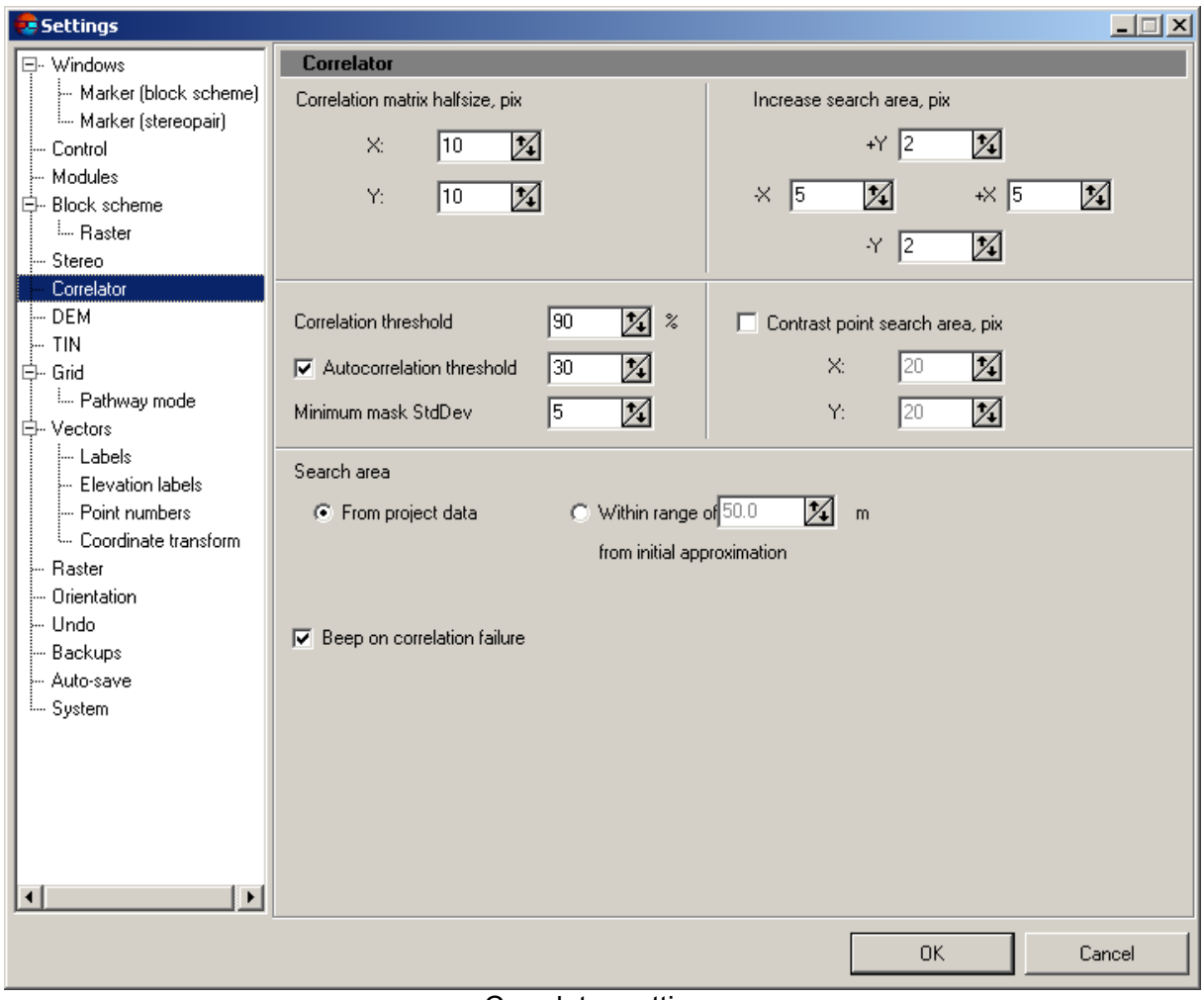
See also chapter Stereomodes.



Stereo mode selection

10.7. *Correlator settings*

For automatic projecting of marker on the terrain surface (see chapter Snap-to-ground mode) **PHOTOMOD** system uses correlation algorithm. The **Correlator** tab of **Settings** window (**Service | Settings** menu command or button) allows to setup the following correlator parameters.



Correlator settings

- **Correlation matrix half-size** – half-sizes of the matrix of searching the corresponding points on the left and right images (the default value is **10x10**). In most cases it is not recommended to increase the matrix size because of slowing down the processing.

- **Increase search area**

  ± X – allows to increase the searching area by X parallax (in pixels). These values may be increased when tie points, used in the relative orientation process, do not "cover" the whole range of Z coordinates for the model (tie points are not located on the very high and very low relief elements).

  ± Y – allows to increase the searching area by Y parallax (in pixels). Usually Y-parallax after the relative orientation is not greater than 1 pixel so in most cases the increasing of this parameter just slows down the process without the results improving.

  - **Correlation threshold** – minimum acceptable correlation value. All points with the correlation less than the selected value are rejected. If you decrease this value the correlator will find more points, but the number of erroneous points also increases. The default value is **90** percent.
  - **Autocorrelation threshold** – for contrast material (e.g. digital imagery of urban area in cloudless weather), this parameter may be set to 8-9; when images have motion blur artifacts or for processing rural area, optimum value is about 30.
  - **Minimum mask StdDev** – defines minimum std. deviation of brightness over the correlation mask (no-contrast, single color surface has std. deviation of zero). There is no direct relationship between this threshold and overall resulting quality. Still, masks with low std.dev. tend to have higher amount of noise relative to real details. In majority of cases it is recommended to leave default value (5). If the imagery has extremely low contrast, the limit may be decreased to 1. If there are enough contrast

objects and correlator produces erroneous points on monotonous areas, it may be worth increasing a bit.

- **Contrast point search area** – defines whether the system should first search for the most contrast point within given range (in pixels). It is recommended to increase this parameter for imagery having objects with contrast edges and decrease for low-contrast images with shadows, where otherwise majority of points would tend to shadow edges, leading to errors.

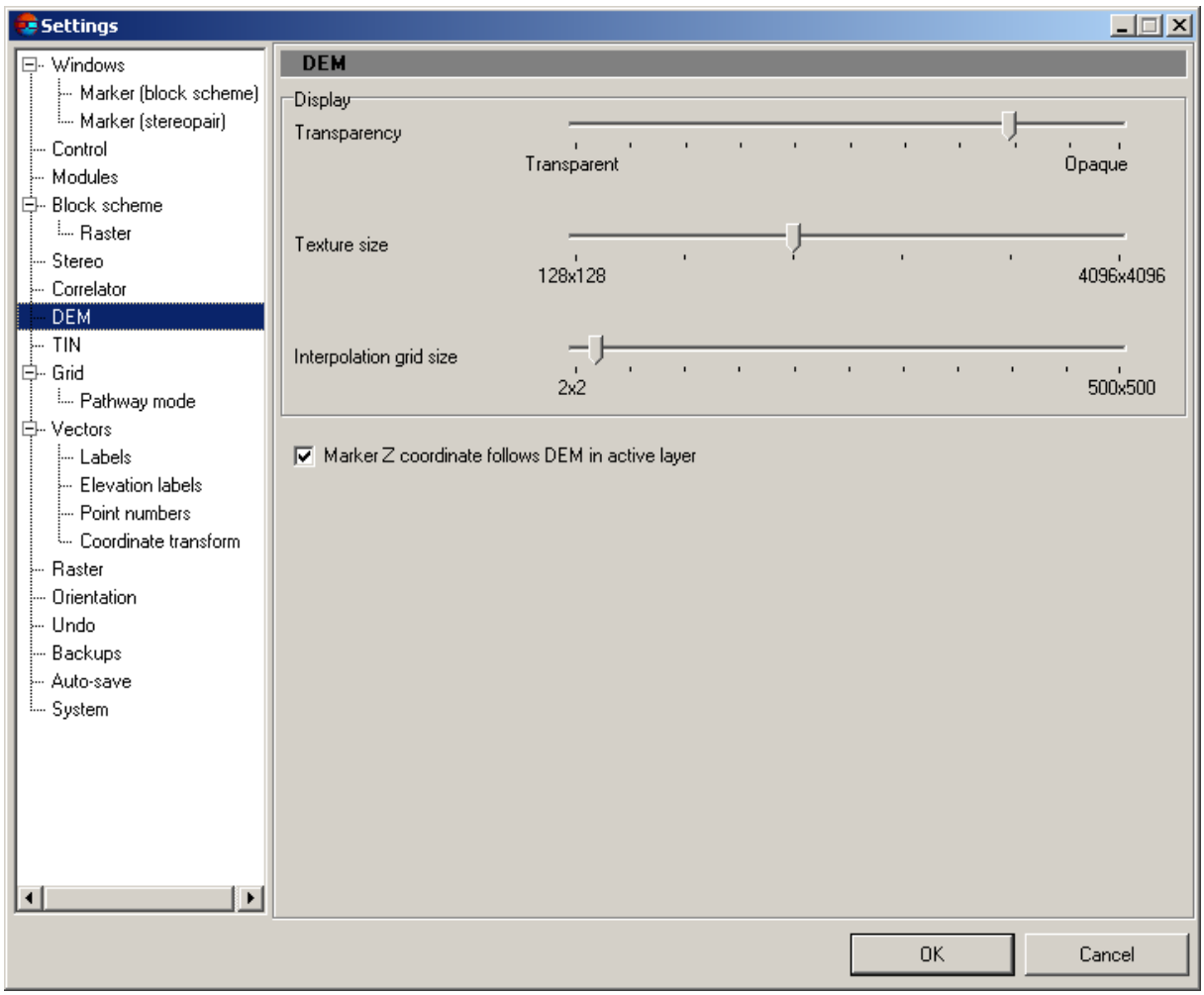
- **Search area** – search area of the appropriate point on the neighbor image:
  From project data – searching will be performed within the range of min and max elevation, measured for each project stereopair.
  Within range of defined value in meters **from initial approximation** – surface searching within the range of defined value from marker position.
- **Beep on correlation failure** – select this option to enable a sound signal (and "bad point" message in the **Status** line), if correlator fails.

10.8. *DEM settings*

The **DEM** tab of **Settings** window (**Service | Settings** menu command or button) allows to setup DEM displaying parameters using the following sliders:

- **Transparency** – allows to setup a level of DEM transparency
- **Texture size** – allows to specify a size of texture used to simulate more or less realistic terrain surface (relief illusion). Moving slider to the right increases detail level and visual quality, but slows down the DEM loading process.
- **Interpolation grid size** – influences on displaying in stereo window (see the chapter Stereomeasurements), allows to define a size of regular grid, that is used for DEM "pulling on" (to comply relief in non-orthogonal projection).

When **Marker Z coordinate follows DEM in active layer** option is on, the marker is settled on DEM surface, i.e. marker Z coordinate is set to corresponding DEM Z in the marker position, if DEM is active layer in Layer manager.

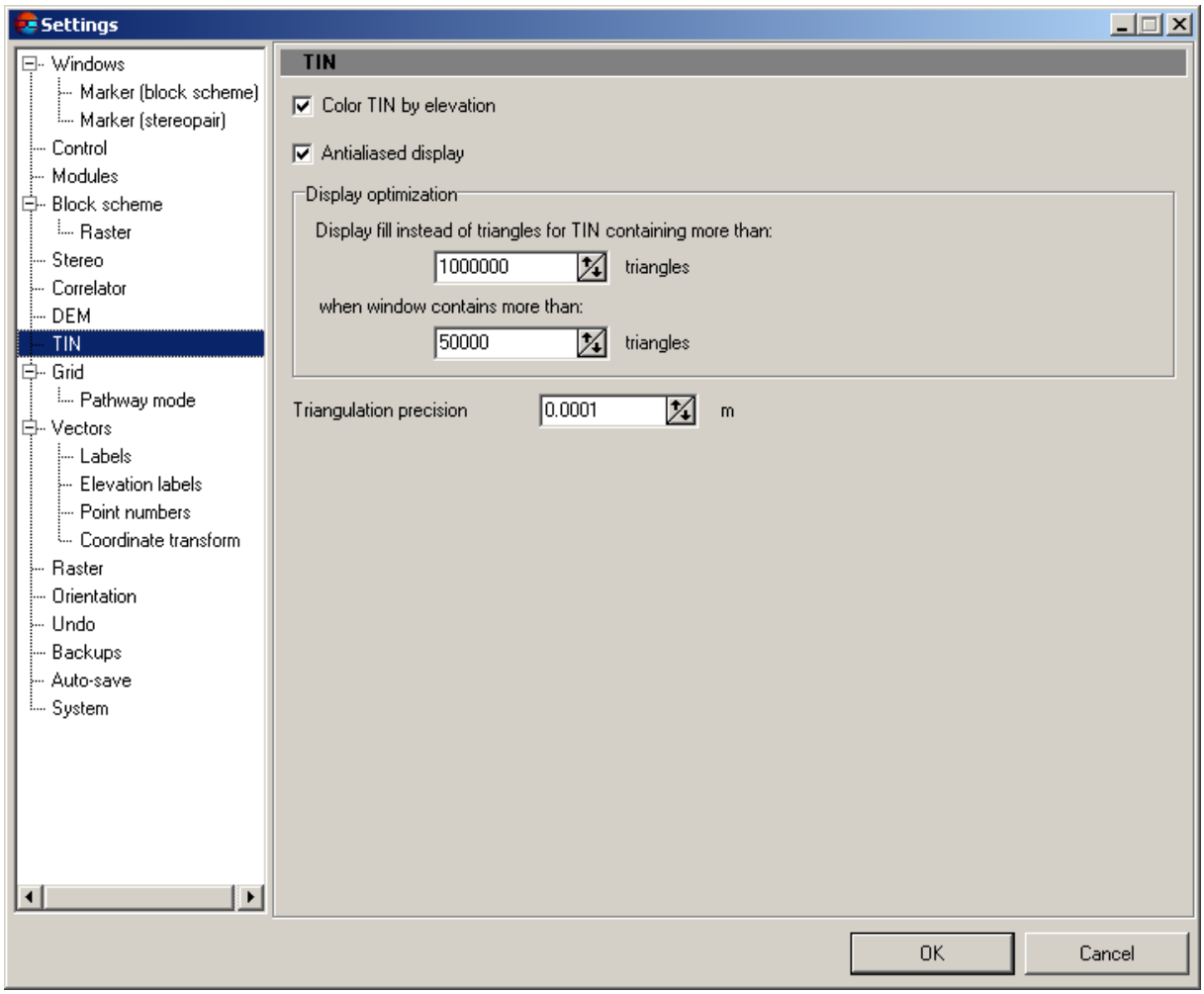


DEM settings

See also the chapter Digital Elevation Model (DEM).

10.9. *TIN settings*

You can setup TIN displaying parameters in 2D window using **Settings** window (opened by **Service | Settings** menu command or button) on the **TIN** tab. See also the chapter TIN.

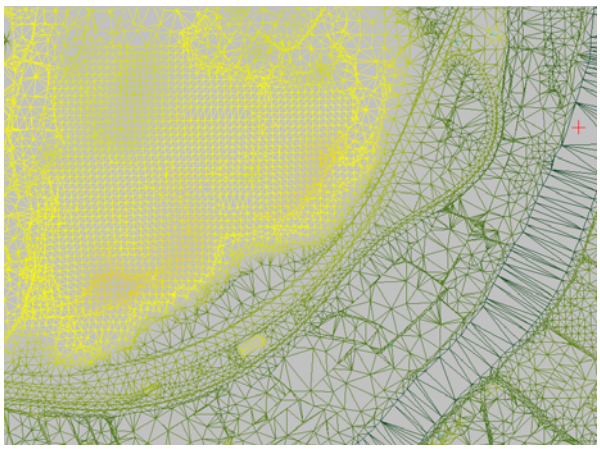


Settings of TIN displaying

The **Color TIN by elevation** option (enabled by default) allows to show TIN as a triangles colored by height or as a solid filling in cases if the TIN contains more than pre-defined number of triangles or when zoom to scale, at which the window contains more number of triangles than specified here.

When displaying TIN as a solid filling, quasi-contours are not shown.

If the **Color TIN by elevation** option is disabled all triangles of TIN are displayed by the same color, defined for the TIN layer in Manager (see also the chapter Layer manager).



TIN as triangles colored by height

TIN as a solid filling

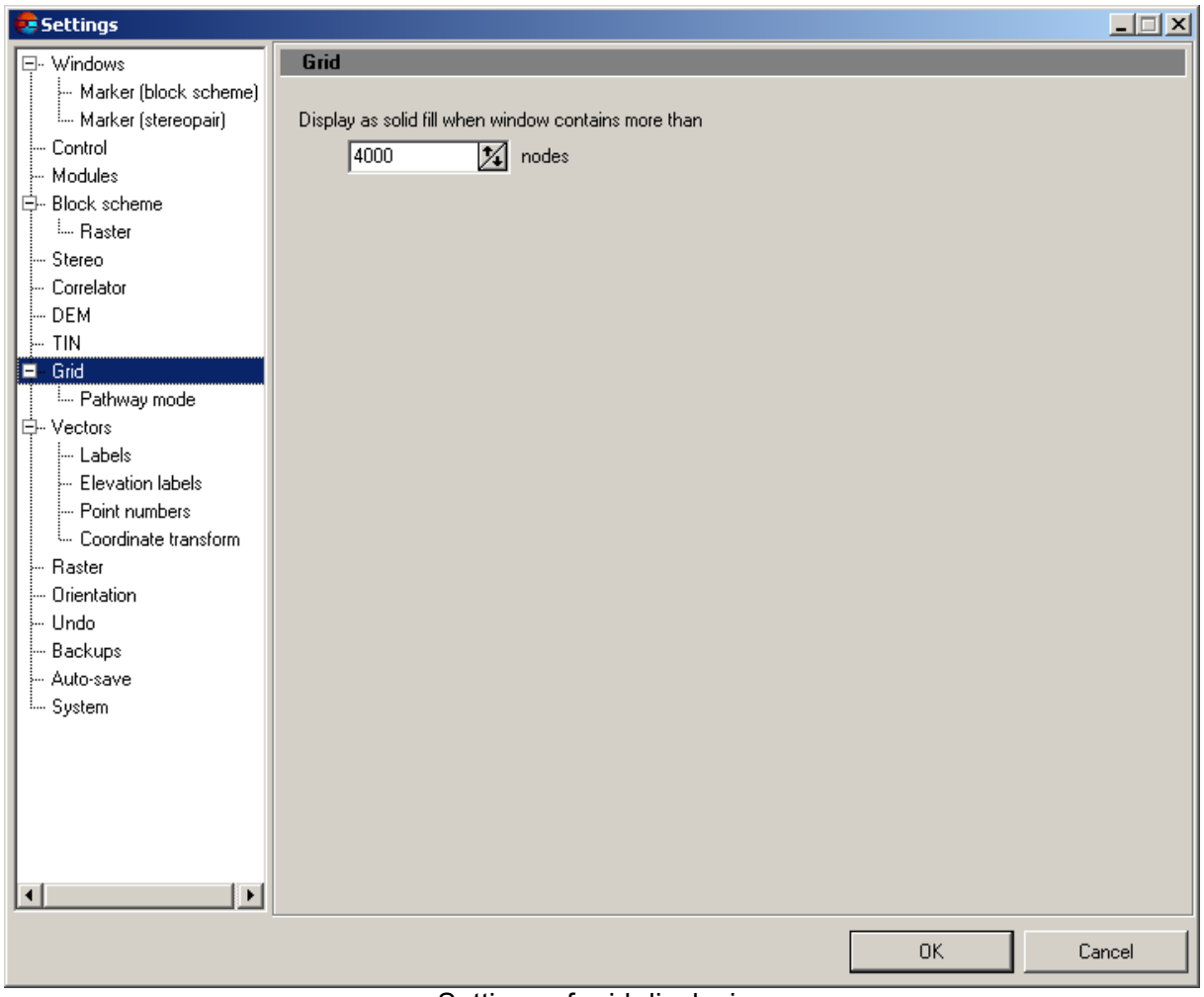
The **Antialiased display** option is used for enhanced visual representation of TIN and quasi-contours.

Changing the **Triangulation precision** may help eliminate TIN creation errors from vectors, containing almost coinciding points. This value defines minimum distance between points which become different TIN nodes, not the single node.

10.10. *Grid settings*

The **Grid** tab of **Settings** window (**Service | Settings** menu command or button) allows setting up parameter of displaying the regular nodes grid (see also chapter Grid creation):

- Display grid as solid fill instead of separate points, if number of nodes in the window at current zoom is more than specified value.
- Set grid node visible size.



Settings of grid displaying

10.11. *Pathway mode settings*

The **Pathway mode** tab of the **Settings** window (**Service | Settings** menu command or button) defines parameters for semiautomatic creating of points in pathway mode.

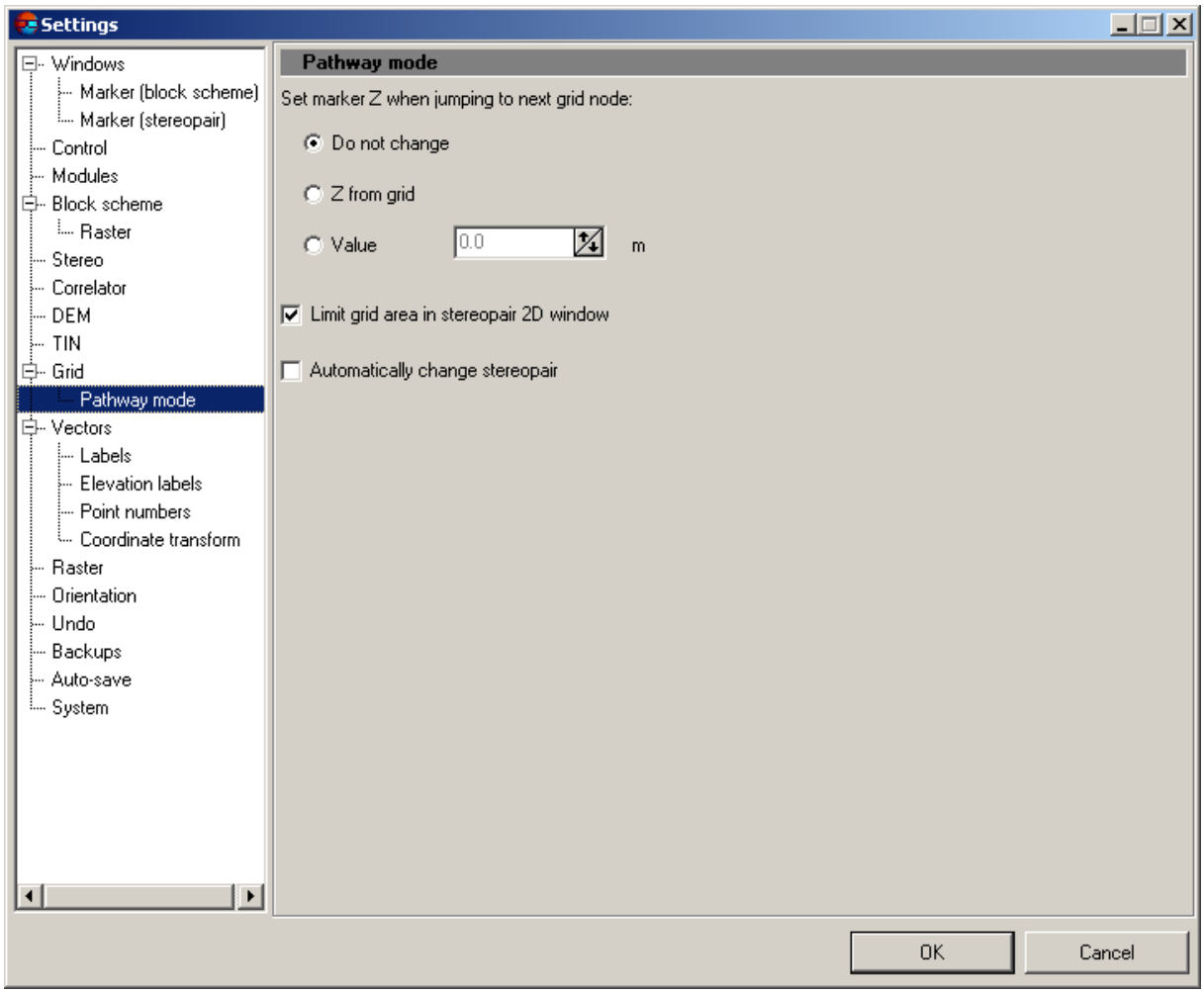
Set marker Z when jumping to next grid node:

- **Do not change**, i.e. Z is left as it was in previous grid node;
- **Z from grid** – equal to Z set in grid properties window (see chapter Grid creation);
- Given **value**.

When the **Limit grid area in stereopair 2D window** option is on, the grid traversing is limited to the stereopair limits, regardless of the total grid size. When this option is on, the **Automatically change stereopair** option should be turned off.

To traverse the "global" grid with changing stereopairs automatically, it is necessary to turn off the **Limit grid area in stereopair 2D window** option and turn on the **Automatically change stereopair** option. At that, automatic stereopair change is affected by the **Threshold** parameter value, defining the border part of stereopair in percent, when reaching which in pathway mode, the stereopair in 2D window is changed. The **Threshold** value is edited on the **Control** tab.

See also chapter Creation of pickets in pathway mode.



Pathway mode settings

10.12. *Vector objects settings*

The **Vectors** tab of the **Settings** window (**Service | Settings** menu command or button) allows settings the following parameters:

- **Default curve smoothing** - from 0 to 1;
- **Points joint swath for topological operations** (relating menu – **Vectors | Topology**). See also chapter Editing of vector objects.
- **Streamline step** (in project units, usually meters). See chapter Streamline mode.

The **Selection** section allows settings the following vector objects selecting options:

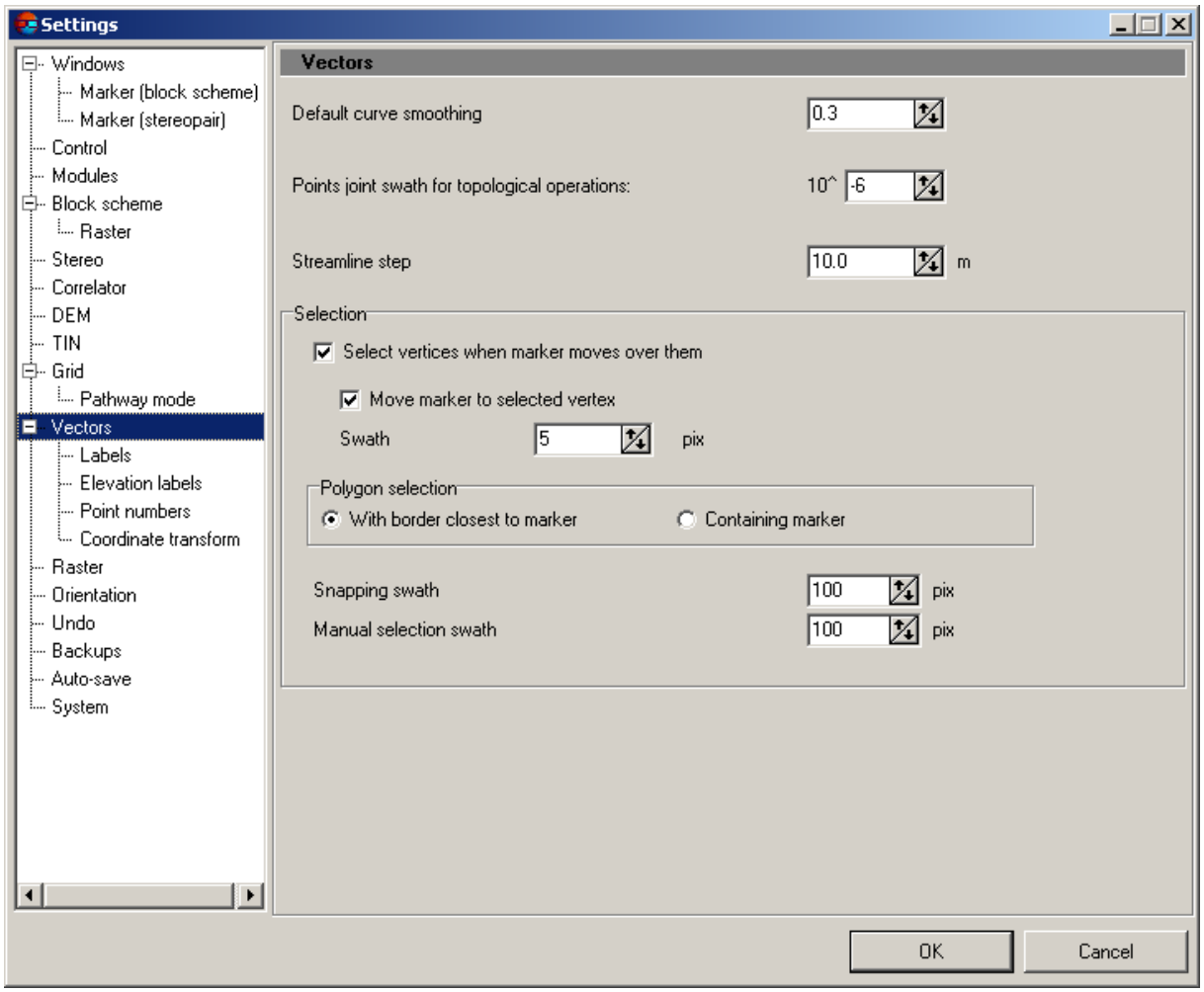
- **Select vertices when marker moves over them** – if on, moving the marker selects point/vertex inside range of **Swath** screen pixels at current zoom.

- **Move marker to selected vertex** – if on, marker moves to the newly selected vertex (thus displaying X,Y,Z coordinates of the vertex in status panel).

The **Polygon selection** subsection allows choosing the polygon selection mode – either polygon with border closest to marker, or polygon, inside which the marker is positioned. See also chapter Objects selection.

Finally, the radius of searching for nearest points/vertices or lines may be specified for snapping (see chapter Type of snapping) or manual selection operations (see chapter Objects selection) with **Snapping swath** and **Manual selection swath** fields respectively.

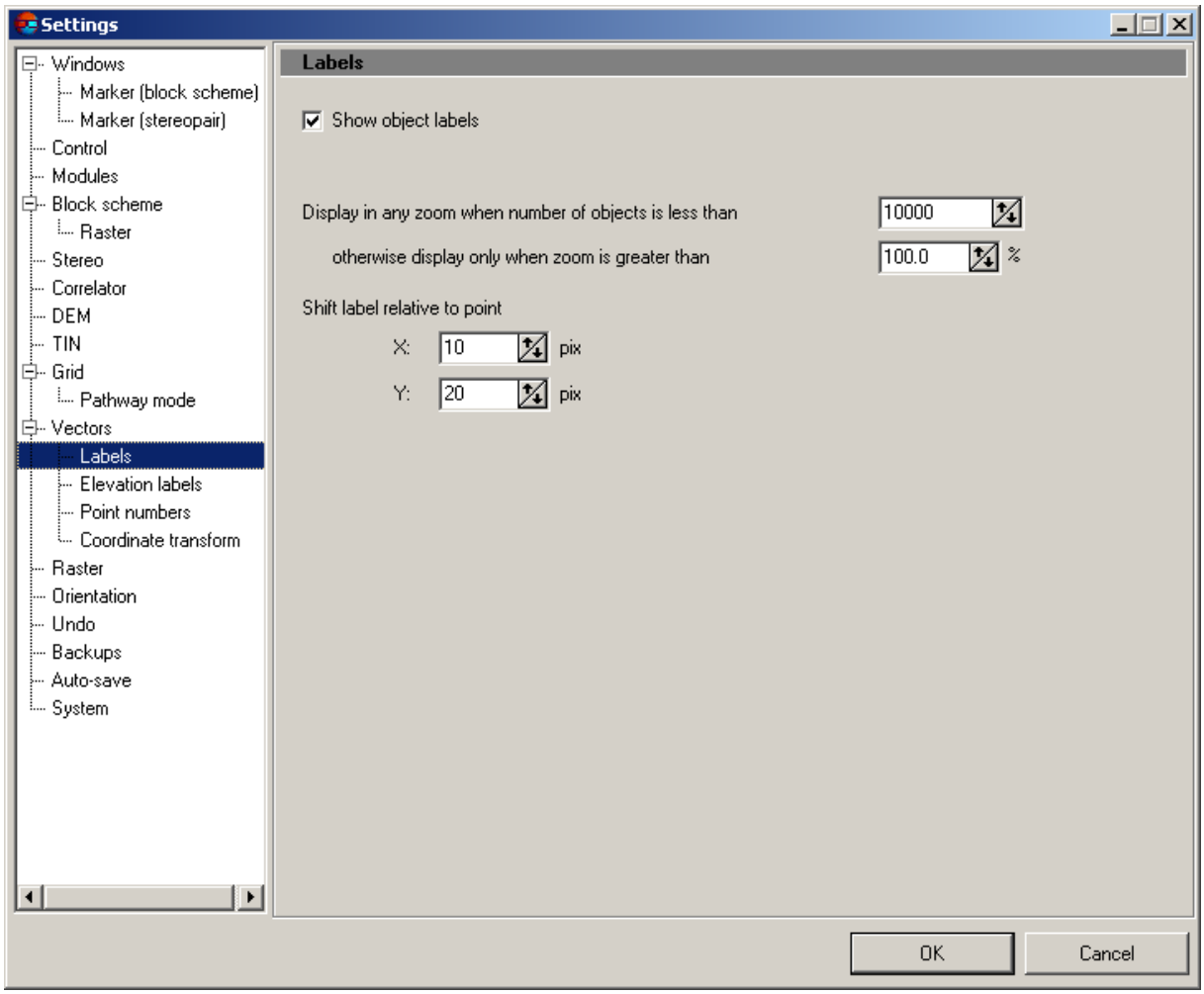
Manual selection swath defines the range around the marker, in which search is performed when manual selection of vector points is done.



Settings of vector objects displaying

10.12.1. Settings of labels displaying

The **Vectors | Labels** tab of **Settings** window (**Service | Settings** menu command or button) is intended for setup of displaying labels that accompany point vector objects or vertices of linear and areal objects. At that a value from **Label** code (from Code table) of the appropriate 3D object will be used as a label. See the chapter Labels creation.



Settings of labels displaying

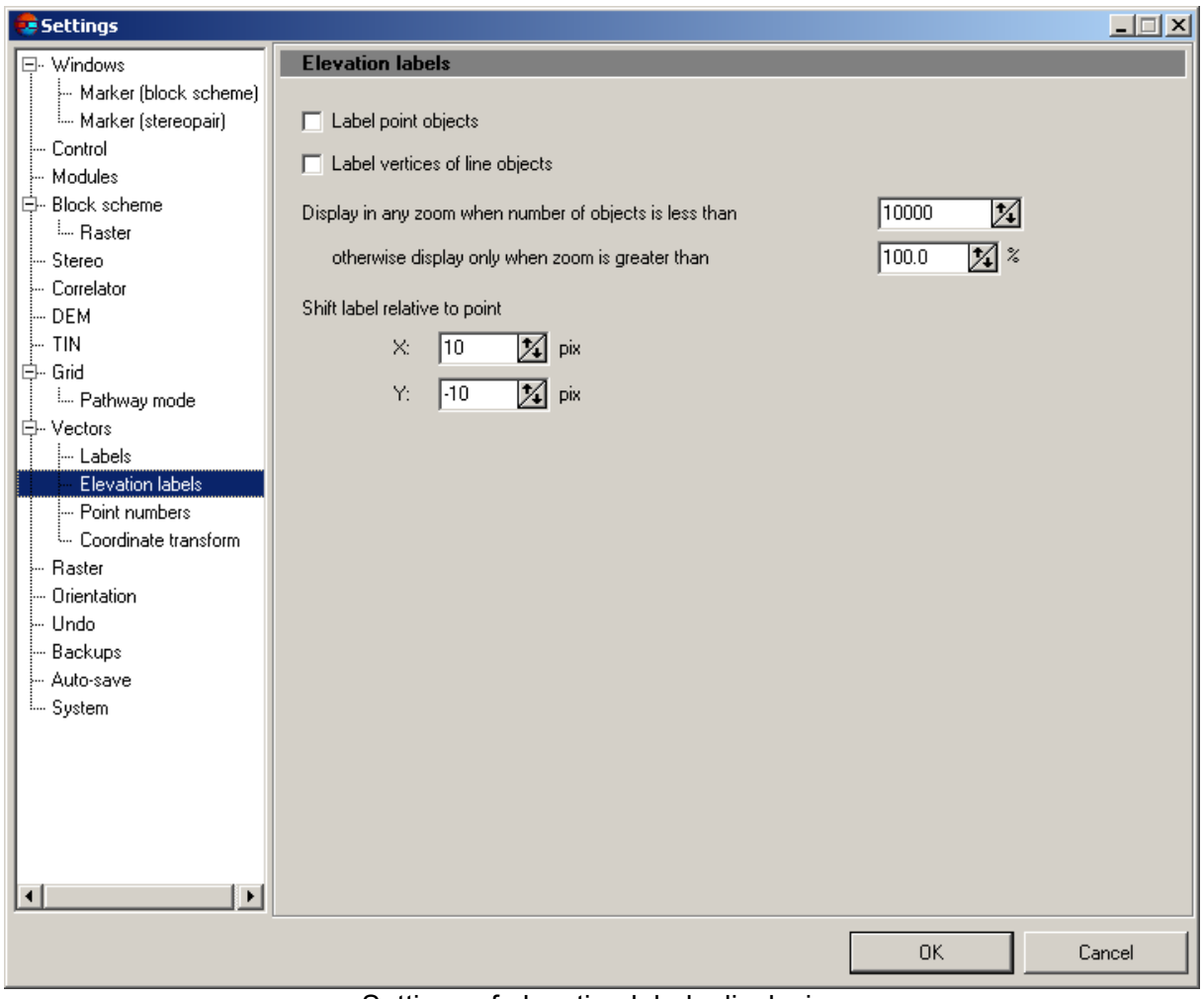
The **Show object labels** option is enabled by default. There are two parameters below it allowing to adjust labels displaying: to display labels if a number of objects at any zoom less then value specified, otherwise if the scale exceeds the specified one.

You can also setup the label shift relatively to the object's center coordinates in pixels by X and Y axes. If offset value is 0, the symbol is placed over the point object.

10.12.2. Settings of elevation labels displaying

The **Vectors | Elevation labels** tab of **Settings** window (**Service | Settings** menu command or button) is intended for setup of displaying elevation labels, that accompany elevation numbers near point objects and/or near vertices of linear objects (see appropriate options, that are disabled by default). There are two parameters below it allowing to adjust elevation labels displaying.

You can also setup the label shift relatively to the object's center coordinates in pixels by X and Y axes. If offset value is 0, the symbol is placed over the point object.

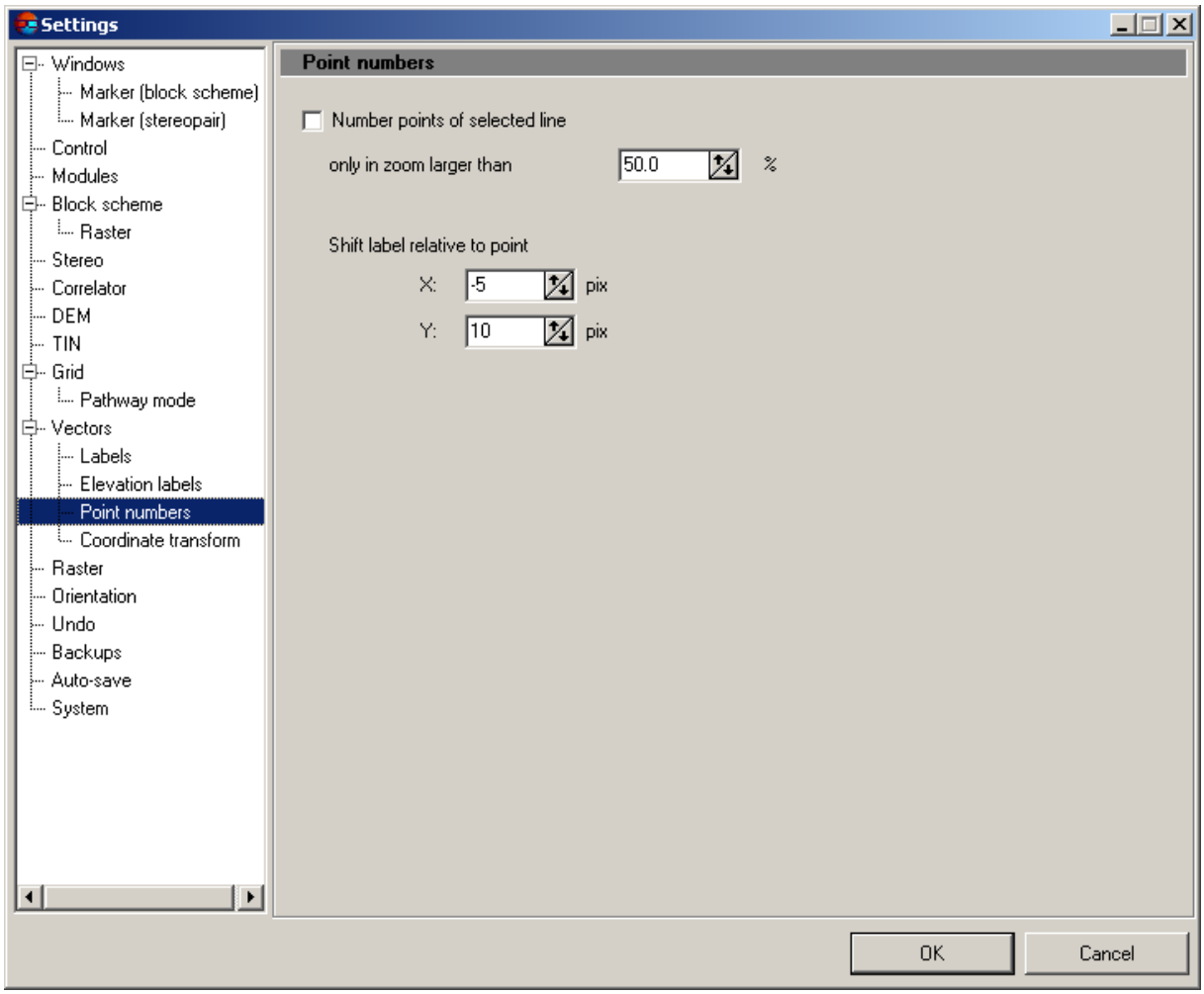


Settings of elevation labels displaying

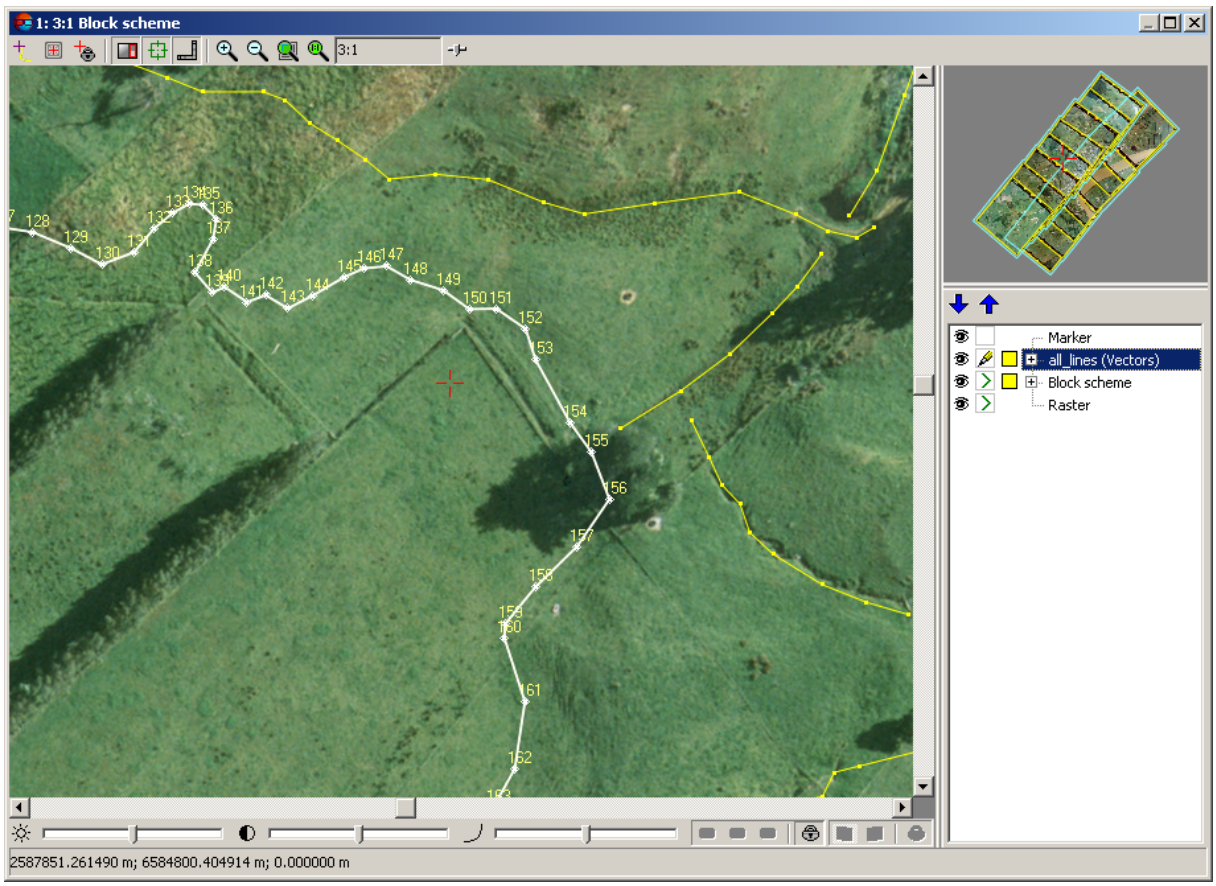
10.12.3. Settings for display of points numbers

The **Vectors | Point numbers** tab of the **Settings** window (**Service | Settings** menu command or button) allows configuring the displaying of point numbers of selected vector object (polyline or polygon). Enable the corresponding option and define settings:

- **Zoom**, starting from which the numbers are shown;
- **Shift** by **X** and **Y** of the number label relative to vector object vertex.



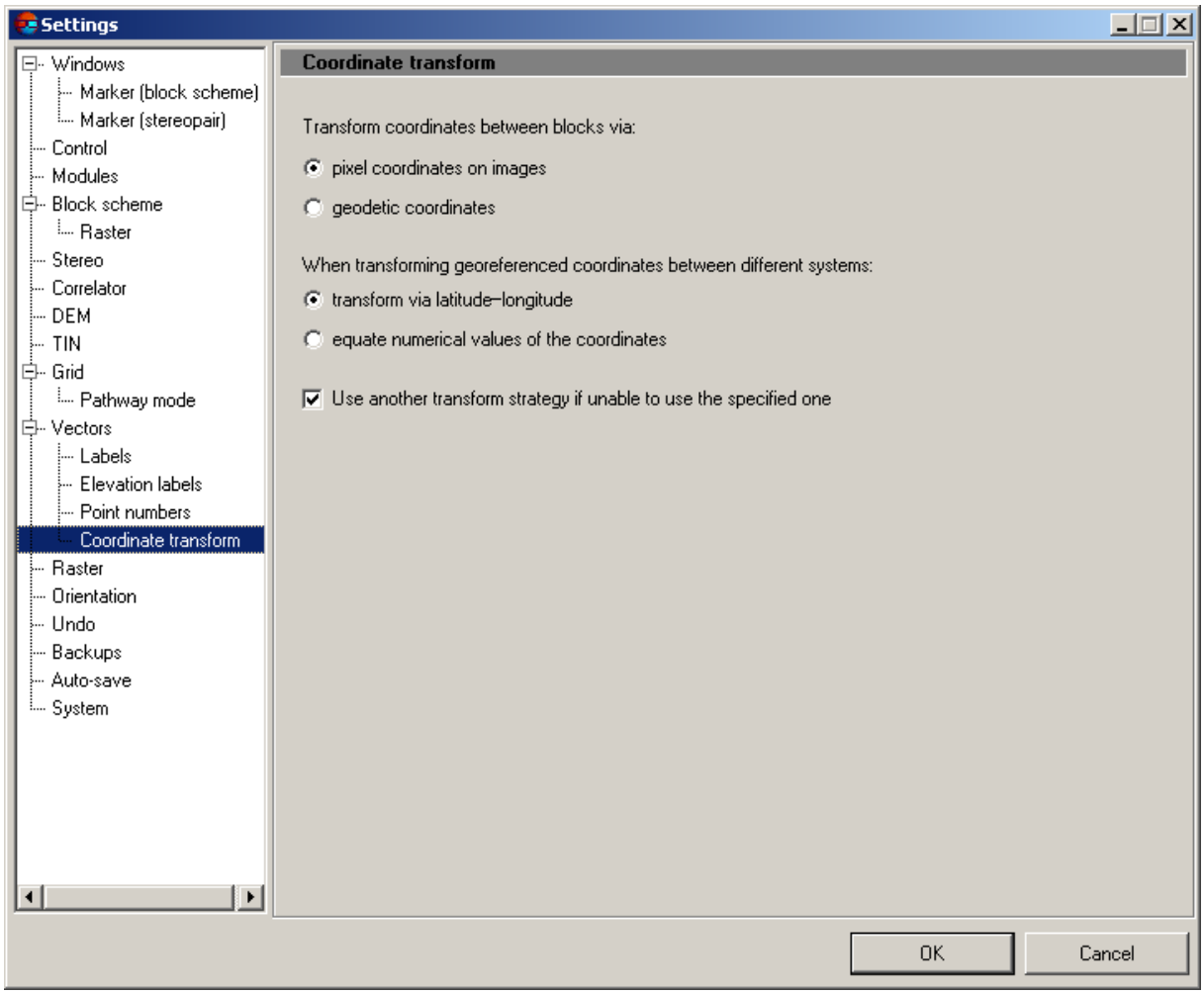
Settings for displaying numbers of selected object vertices



Display of vector objects vertices numbers in 2D window

10.12.4. Coordinates transformation

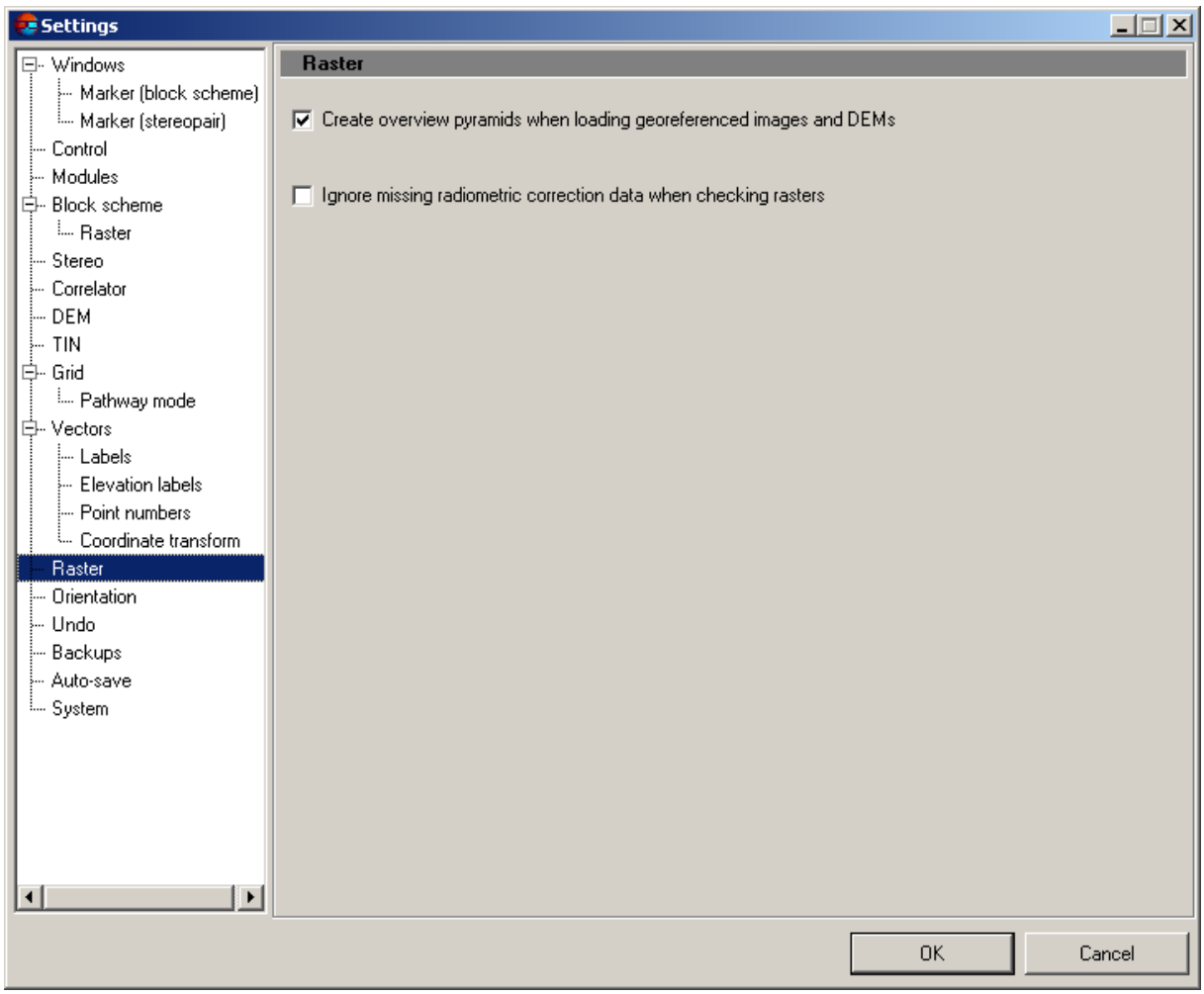
The **Vectors | Coordinates transform** tab of **Settings** window (**Service | Settings** menu command or button) allows to perform coordinates transformation between blocks through pixel or geodetic coordinates. In case of using geodetic coordinates, you can perform transformation via latitude/longitude, or using equating of coordinates numerical values. The option below allows to use another strategy of coordinates transformation.



Strategies of coordinate transformation

10.13. *Parameters of images and DEM loading*

The **Raster** tab of **Settings** window (**Service | Settings** menu command or button) allows to setup a parameter of pyramids creation during loading of georeferenced images, and DEM's (the option is enabled by default). Availability of created pyramid influences on speed of images refreshing.
The option below allows to switch off the images checking for radiometric correction (without ImageWizard).



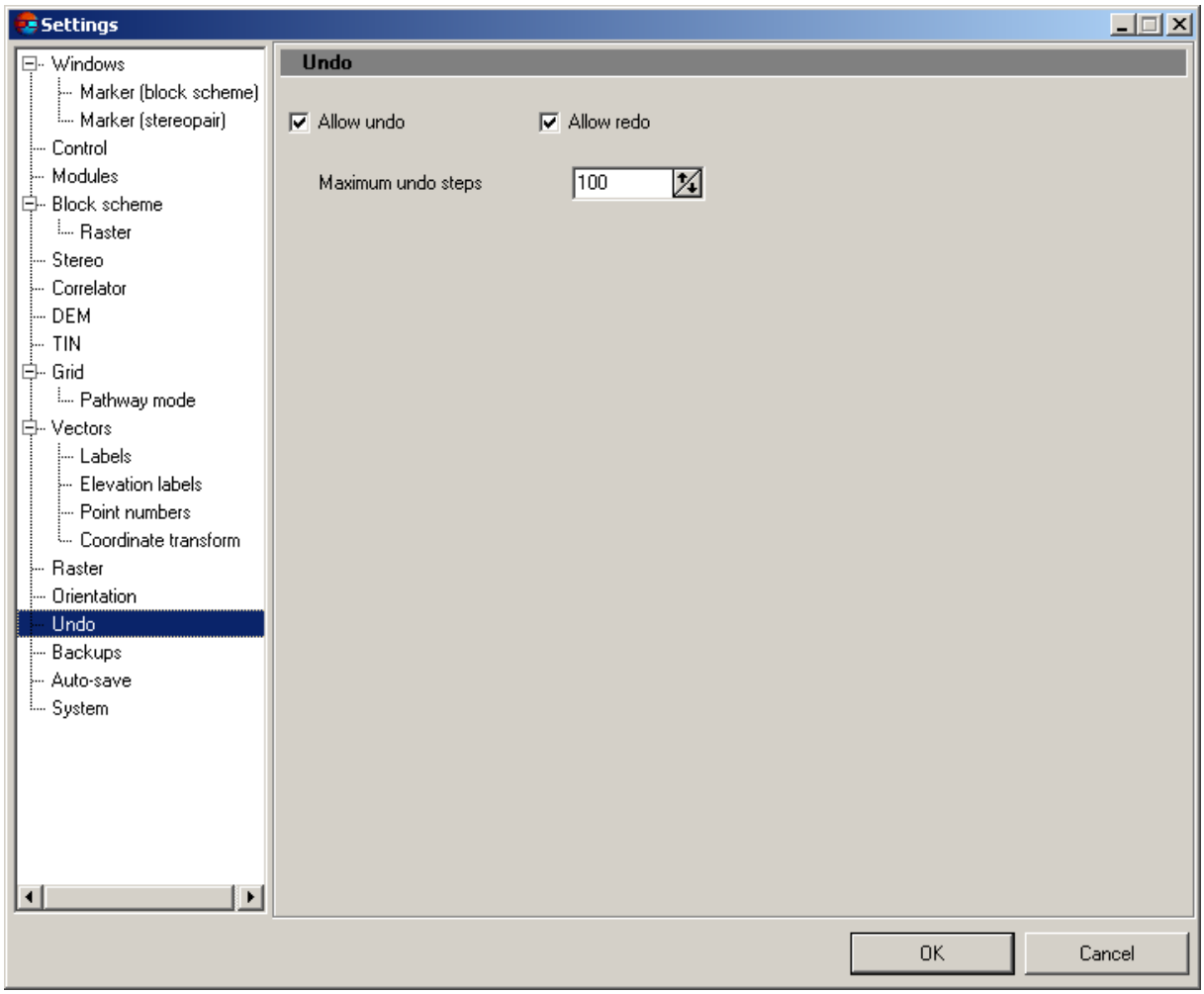
Parameter of images loading

10.14. *Orientation settings*

The **Orientation** tab in the **Settings** window (**Service | Settings** menu command or button) allows to set whether all images are selected automatically for automatic tie points measurements when the corresponding window is opened; otherwise only those images selected in the block layout 2D window are chosen.

10.15. *Undo settings*

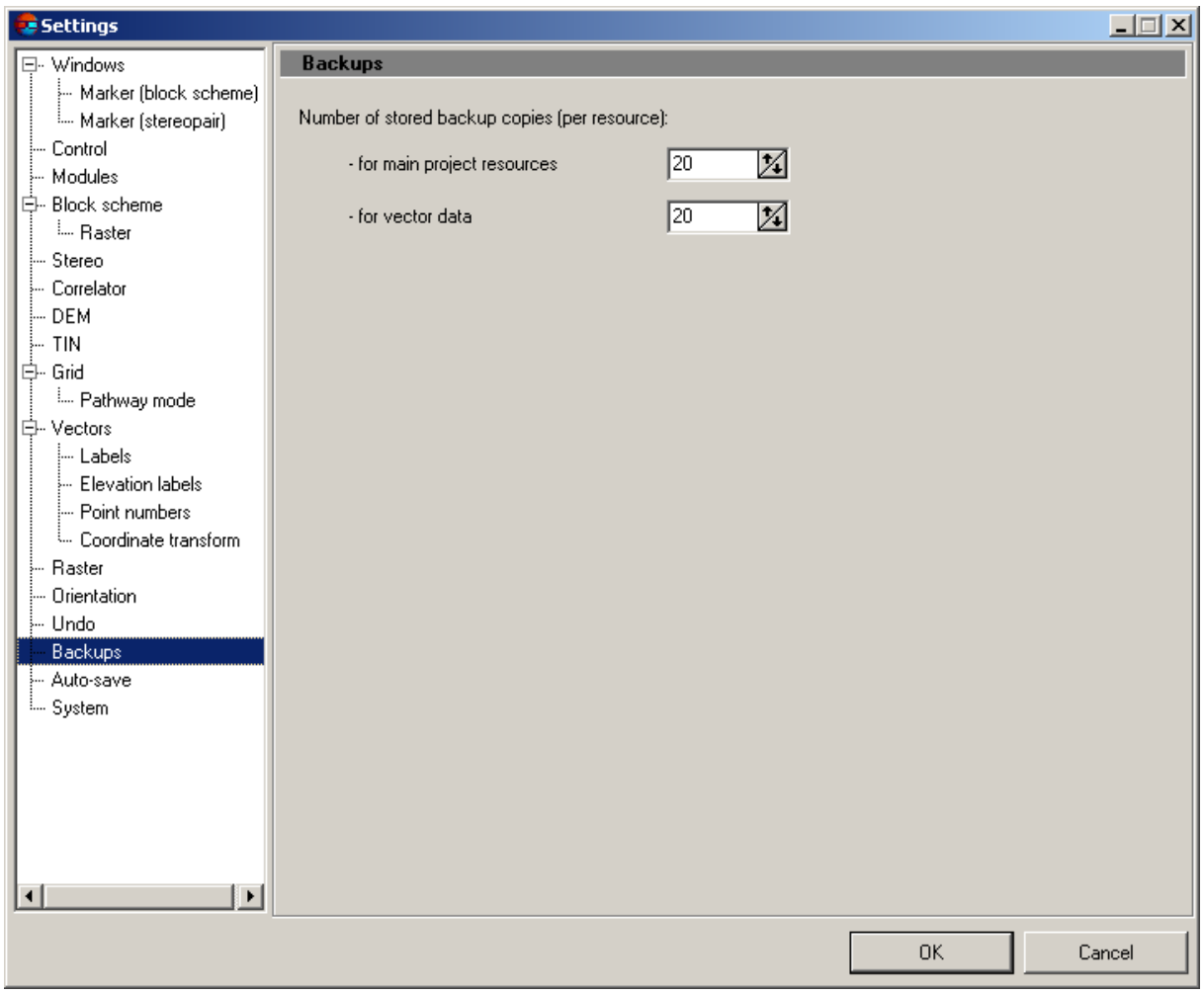
The **Undo** tab of **Settings** window (**Service | Settings** menu command or button) is used to enable or disable undo/redo functionality and to set up the depth of undo actions stack (stored number of undo/redo actions).
See the chapter Undo.



Undo settings

10.16. *Settings for resource backups*

The **Backups** tab of the **Settings** window (**Service | Settings** menu command or button) allows defining the number of stored backup copies of primary project resources and vector resource. When the numbers set here are exceeded, the oldest backup copies are deleted. The backup copies are stored in the **backup** subfolder of the folder containing the resource being saved (e.g. for *data/vectors.x-data* resource, the backup copies are placed in the *data/backup* subfolder).



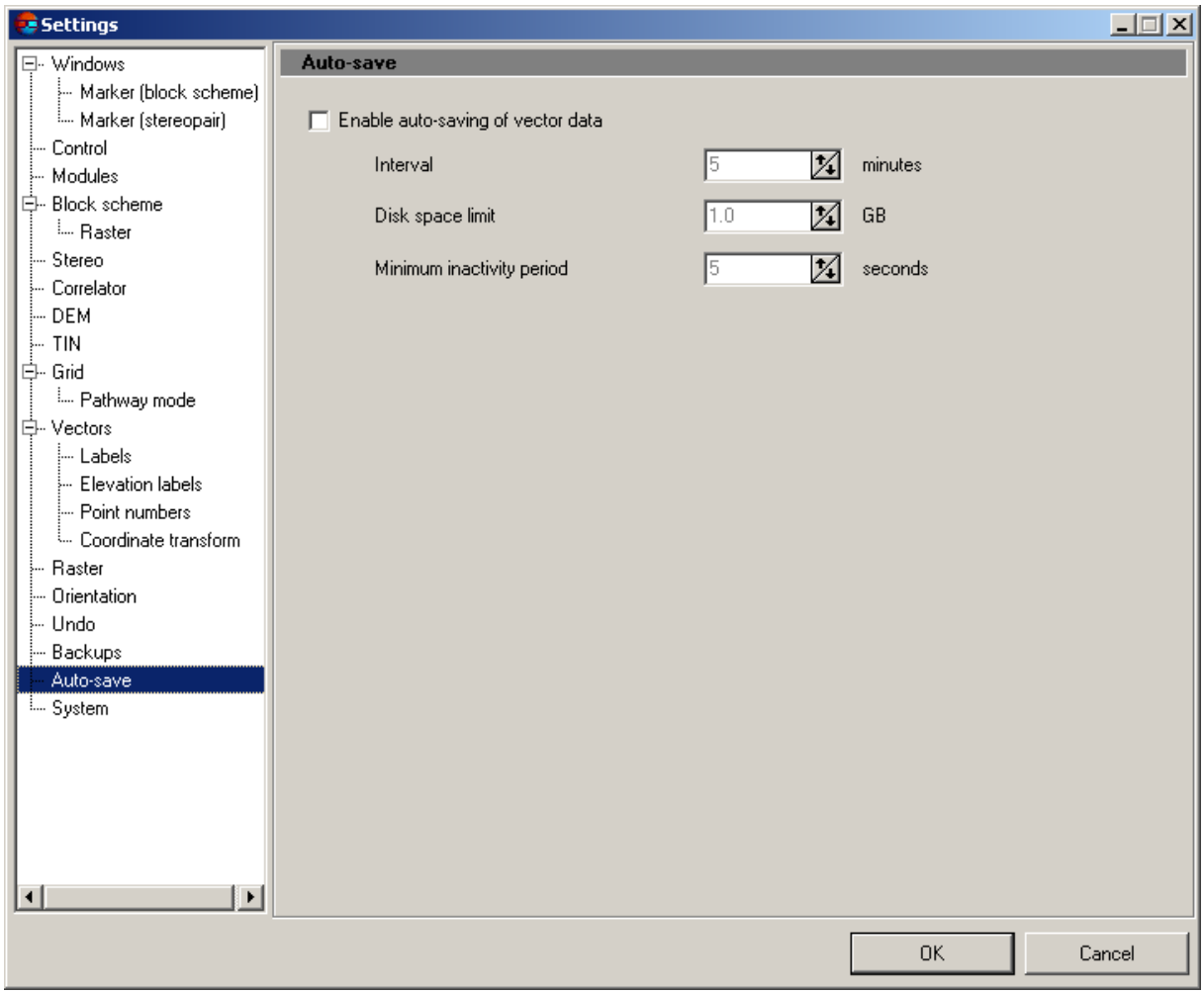
Resource backups settings

10.17. *Auto-save settings*

The **Auto-save** tab in the **Settings** window (**Service | Settings** menu command or button) allows configuring periodic automatic saving of vector data being edited. Turn on the corresponding option and set the following auto-save parameters:

- **Interval** (in minutes) between consequent auto-saving of edited objects (in case this option is disabled, see the option below)
- **Disk space limit** (in GB) which is used for auto-save data (in PHOTOMOD5.VAR\AutoSave folder). When this limit is exceeded, the oldest data files are deleted.
- **Minimum inactivity period** (in seconds) – minimum system idle time, after which the auto-save is performed if Interval minutes have lapsed since the previous auto-save. This function ensures that auto-saving is executed when there is no mouse or keyboard activity, so that this does not bring any discomfort to operator.

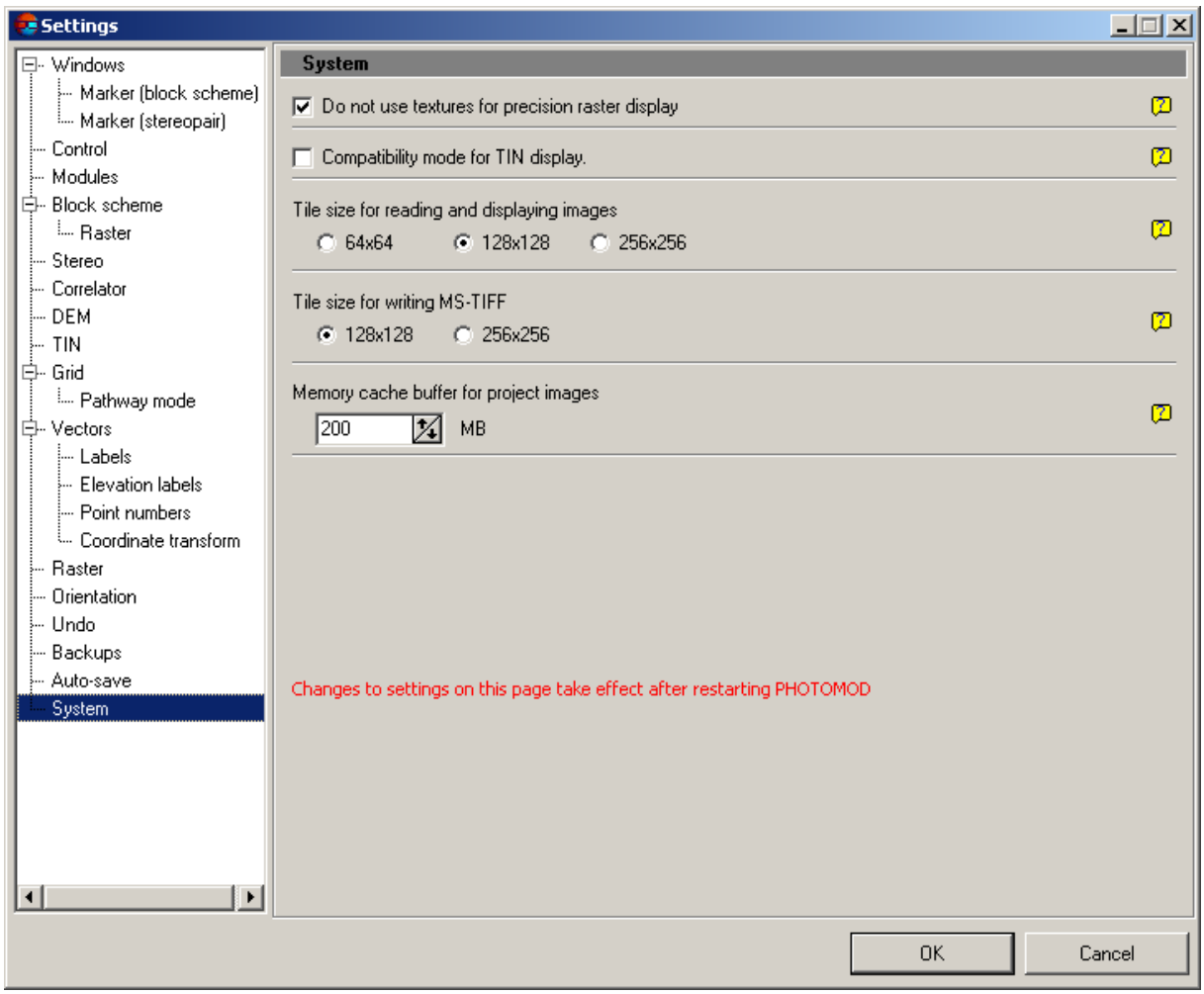
The vector data files are stored in **PHOTOMOD5.VAR \ AutoSave** folder. Resource name with appended date is used as the file name. Autosave is activated only after the layer has been modified (and it's name marked with an asterisk in the layer manager window). When the set limits of disk space is exceeded by the autosave files, the oldest files are automatically erased on **PHOTOMOD Core** start or autosave initiation. The **Vectors | Restore** menu command opens the **Restore files** window listing the contents of the **Autosave** folder and allowing to load any of these files.



10.18. *System settings*

The **System** tab of **Settings** window (**Service | Settings** menu command or button) allows to setup system settings:

- **Do not use textures for precision raster display** – enable this option to speed up graphics output when using old video adapter, but this will not allow to position marker with sub-pixel accuracy (e.g., in fixed marker mode).
- **Compatibility mode for TIN display** – enable this option only if you can see artifacts during TIN displaying.
- **Tile size for reading and displaying images** – try to change a size of tile, if during image displaying in 2D window you experience noticeable delays.
- **Tile size for writing MS-TIFF** – allows to setup a tile size for files of MS-TIFF and MegaTIFF formats to be created.
- **Memory cash buffer for project images** – when working with big volume of vector data, reduce memory cash buffer (free memory), in other cases increase it.



System settings

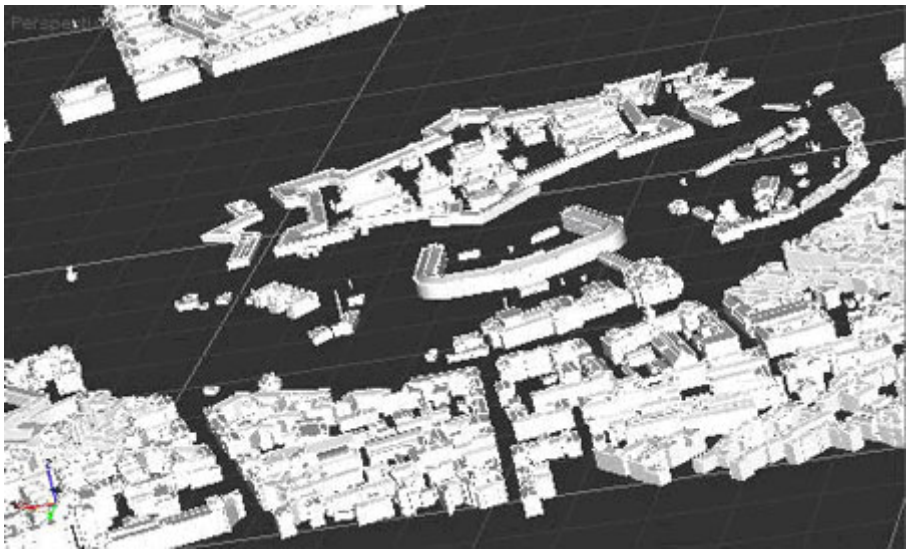
10.19. *Mouse settings*

The time consuming processes such as break lines vectorization or TIN editing requires the system to be as ergonomic as possible. Besides using hot keys (see the chapter **Hot keys**) and toolbar icons instead of the main menu, there is a very useful capability to setup mouse buttons for different operations. At that you can use either ordinary 3-button or 5-button mice or special devices: 3D mice, hand wheels and foot pedals. In fact you can setup any mouse which emulates the standard 5-button one. To setup the mouse use menu command **Service | Mouse setup** to open **Mouse settings** window. See **General information** User Guide for details of mouse configuration and special devices and 3D mice usage.

11. Three-dimensional modeling – 3D-Mod

3D-MOD module is intended for building of 3D-objects using vectors with classifier created in PHOTOMOD system (see the chapter Vector layer with classifier), editing, texture mapping and for subsequent export of resulting file to the necessary data format for further use in other software. There is the following workflow:

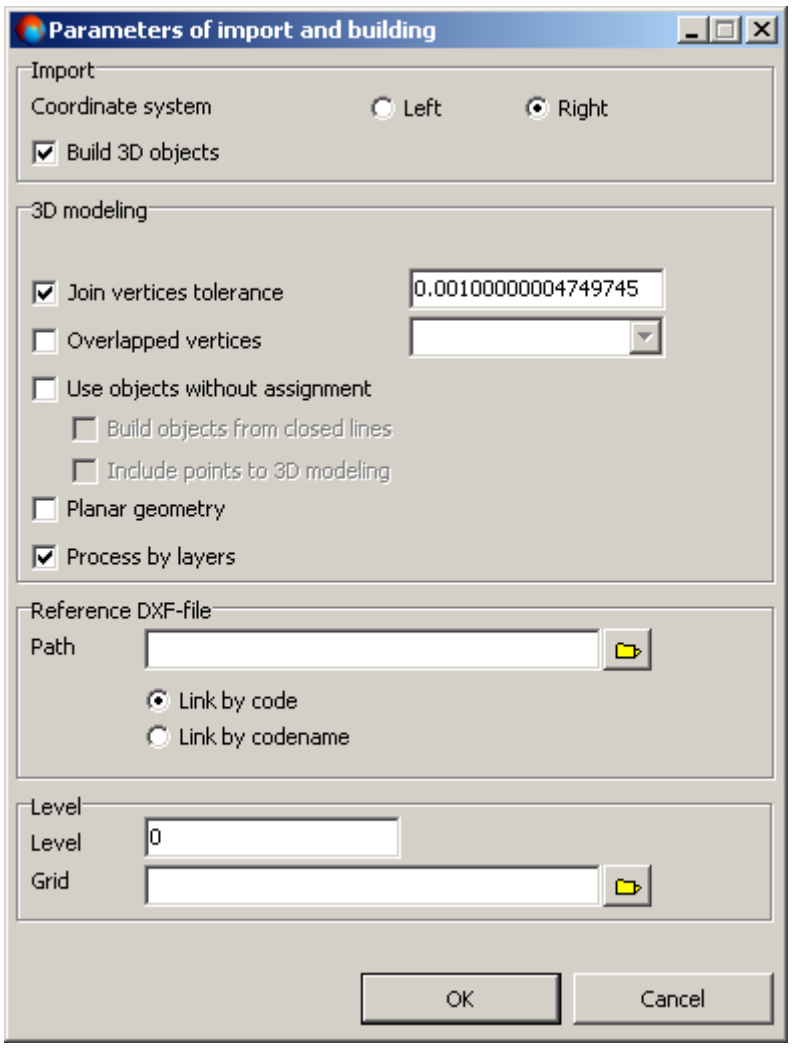
- Export vectors with attributes to ASCII-A format (see the chapter Export to ASCII-A);
- Start 3D-Mod program;
- Import of vectors from ASCII-A-file and transforming them to the 3D-Mod native format (*.tx3);
- 3D modeling (including the editing and texture mapping of 3D objects);
- Export to the necessary data format.



11.1. *Main menu of 3D-Mod program*

The **File** menu is used for import of vector objects from ASCII-A-file to 3D-Mod and export of resulting objects of 3D-Mod system to the necessary data format for further use in other software. Menu consists of the following items:

- **Open** – opens the file of vector objects in the 3D-Mod native format (*. tx3).
- **Save** – allows saving the current file or changes of the current file to the 3D-Mod native format.
- **Save as** – allows saving the current file or changes of the current file under a new name in the 3D-Mod native format.
- **Import** – allows to import vector objects from ASCII-A-file and to transform them to native format of 3D-Mod. The command opens a window for ASCII-A-file (*.txt) selection. Format ASCII-A allows to store vector objects attributes along with their metrics (see Appendix B). After selection of .txt file, push the **Open** button to specify parameters in the **Parameters of import and building** window:



Import parameters

The **Parameters of import and building** window consists of 4 sections, where you can specify settings of coordinate system (which is defined in PHOTOMOD project), setup parameters of objects building, specify reference DXF-file (path to DXF-file that contains a description of point objects geometry) and level:

Import section:

- **Coordsystem** – specify **Left** or **Right** coordinate system according to parameters of coordinate system defined in **PHOTOMOD** project;
- **Build 3D objects** – enable this option to build 3D objects;

3D modeling section:

- **Join vertices tolerance** – allows to merge vertices, if the distance between them is less than specified value;
- **Overlapped vertices** mode sets the processing rule for vertices with the same XY during the 3D-models building;
- **Use objects without assignment** – allows to use lines (**Build objects from closed lines**) and contours (**Include points to 3D modeling**) without attributes, just as a bounding contours;
- **Planar geometry** – enables algorithm used to eliminate self-crossing of vector objects and bounding contours in XY plane.
- **Process by layers** mode allows to process elements (closed contours, lines) during the 3D-model building independently for each layer, to remove unwanted interaction between elements from different layers and accelerate processing;

Reference DXF-file section:

- **Link by code** – allows to link point objects with geometry containing in DXF-file by code of object (see Appendix B).
- **Link by codename** – allows to link point objects with geometry containing in DXF-file by name of object code (see Appendix B).
- **Path** – allows to specify path to DXF-file that contains a description of point objects geometry. Push the button to specify a path, after that a dialogue for DXF-file selection is opened. Once DXF-file is selected, push the **Accept** button to perform import.

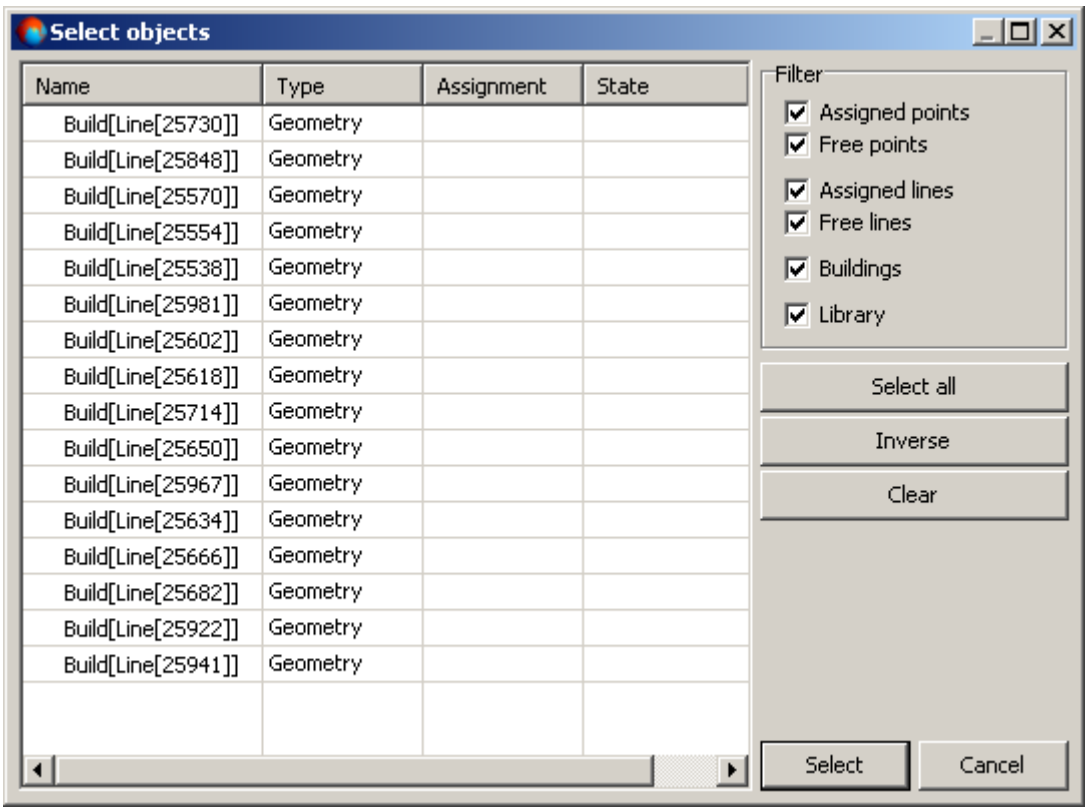
Level section:

- **Level** – set the level of height, on which the side faces of objects will be projected;
- **Grid** – specifies path to the DEM, which sets a lower limit during the 3D objects creating.

- **Export** – used to export objects of 3D-MOD system to the necessary data format for further use in other software. To perform data export to **DXF** format (exchange format of **AutoCAD** system) select menu command **File | Export**, resulting in opening of the **Export** window which allows to create output file with selected extension (*.dxf, *.txt, *.tx3, *.dae, *.3ds).
- **Export selected** – used to export selected objects of 3D-MOD system to selected format – opens the window for format file selection (**Export**).
- **Close** – closes the current file with the ability of data saving.
- **Exit** – closes 3D-Mod program.

The **Edit** menu is used for work with objects imported from **PHOTOMOD** system. Menu consists of the following items:

- **Select** – (is duplicated by pushing the button on toolbar or by **Ctrl-S** shortcut) – allows to select objects in view area.
- **Select by name** – (is duplicated by pushing the button on toolbar or by **Ctrl-N** shortcut) – allows to select objects in view area using a list. After selecting the command the **Select objects** window is opened. It contains a list of names shown as a table, the **Filter** panel and three buttons below the panel, used for managing of objects list.



The list of objects names of 3D-MOD system

The list of names contains the following columns:
Name – unique name with length not exceeding 64 symbols, which is assigned to each object;
Type – object type. May have following values: Geometry, Contour, Line, Point.
Assignment – attribute, that defines a way of using of imported objects during creating 3D-objects (**LibPoint**, **Break**, **Object**).
State – **Used**/**Free** – status of imported objects during 3D-objects creation – used/not used.

The **Filter** panel is used to select the following filters:
Assigned points/Free points –
Assigned lines/Free lines –
Buildings – 3D-objects created using objects imported from StereoDraw module.
Library – 3D-objects created using point objects imported from StereoDraw module and DXF-file that contains a description of objects geometry.
There are the following buttons that allow to manage the objects list:

- **Select all** – selects all items in the list;
- **Inverse** – inverts selection (all selected objects become deselected and all deselected objects become selected);
- **Clear** – cancels selection of all items of the list;

When you selected all necessary filters, click the **Select** button.

- **Select dependent** menu command (is duplicated by pushing the button on toolbar) allows to select both the whole object and all elements (contours, breaklines), used to create objects in 3D-MOD system.

- **Select unused objects** menu command (is duplicated by pushing the button on toolbar) allows to select elements (contours, breaklines), that are not used to create objects in 3D-MOD system.

- **Edit texture coordinates** menu command activates the editing mode of the texture coordinates of 3D objects (see chapter **Texture mapping to 3D objects**).
- **New material** menu command allows specifying the unique texture for the selected object.
  Some objects of 3D scenes can have the common texture. Changing the texture of one of the objects makes it automatically changed for the other objects. To define the unique texture for selected object of 3D scene it's necessary previously to define the *new material* for it.
- **Edit** menu command activates the editing mode of vertices, edges and faces of 3D object (see chapter **Objects editing in 3D Mod**).
- **Delete** menu command (is duplicated by the **Del** key) – deletes selected objects.
- **Delete all** menu command – deletes all objects.

The **View** menu is used to manage view area. It contains the following items:

- **Tools** shows/hides the following toolbars:
  - **View control tools** - shows/hides the toolbar , used to manage visualization area and to stereo mode selection.
  - **Objects control tools** - shows/hides the toolbar , intended for objects management.
  - **Object hidden tolls** - shows/hides the toolbar , intended to manage visualization mode of different objects types.
- **Move** - (is duplicated by pushing the button on toolbar or by **Ctrl-Alt-M** shortcut) – allows to move view area by mouse using drag-and-drop mode.
- **Approach/Distance** - (is duplicated by pushing the button on toolbar or by **Ctrl-Alt-F** shortcut) – allows to move view area closer or farther from user.
- **Rotate** - (is duplicated by pushing the button on toolbar or by **Ctrl-Alt-R** shortcut) – allows to rotate view area by mouse with pressed left mouse button.
- **Zoom** - (is duplicated by pushing the button on toolbar or by **Ctrl-Z** shortcut) – allows to change view area scale.
- **Select region** - (is duplicated by pushing the button on toolbar or by **Ctrl-Alt-S** shortcut) – defines borders of view area.
- **Edges** – viewing 3D objects in a wireframe model.
- **Textures** – viewing 3D objects with the mapped texture.
- **Anaglyph stereo** – enables/disables anaglyph stereomode.
- **Frame stereo** – enables/disables page-flipping stereomode.
- The **View direction** option allows to specify 6 basic orthogonal and one perspective projection (**Front, Back, Left, Right, Top, Bottom, Perspective**).
- **Four views** – (is duplicated by pushing the button on toolbar) allows to see view area in four projections.

The **Objects** menu is used to enable or disable visualization of specific sets of objects and contains the following items:

- **Show points** menu command (is duplicated by pushing the button on toolbar) shows/hides all points.
- **Show lines** menu command (is duplicated by pushing the button on toolbar) shows/hides all points lines and contours.
- **Show buildings** menu command (is duplicated by pushing the button on toolbar) shows/hides all buildings.
- **Show library** menu command (is duplicated by pushing the button on toolbar) shows/hides all objects containing in DXF-file, which is added during import of data from PHOTOMOD system.
- **Hide selected** menu command (is duplicated by pushing the button on toolbar) allows to hide selected objects.
- **Hide by name** menu command (is duplicated by pushing the button on toolbar) allows to hide objects selected by name.
- **Hide unselected** menu command (is duplicated by pushing the button on toolbar) allows to show selected objects.
- **Unhide by name** menu command (is duplicated by pushing the button on toolbar) allows to show objects selected by name.
- **Unhide all** menu command (is duplicated by pushing the button on toolbar) shows all objects.
- **Build** menu command allows building 3D objects by selected vectors.

11.2. *Texture mapping to 3D objects*

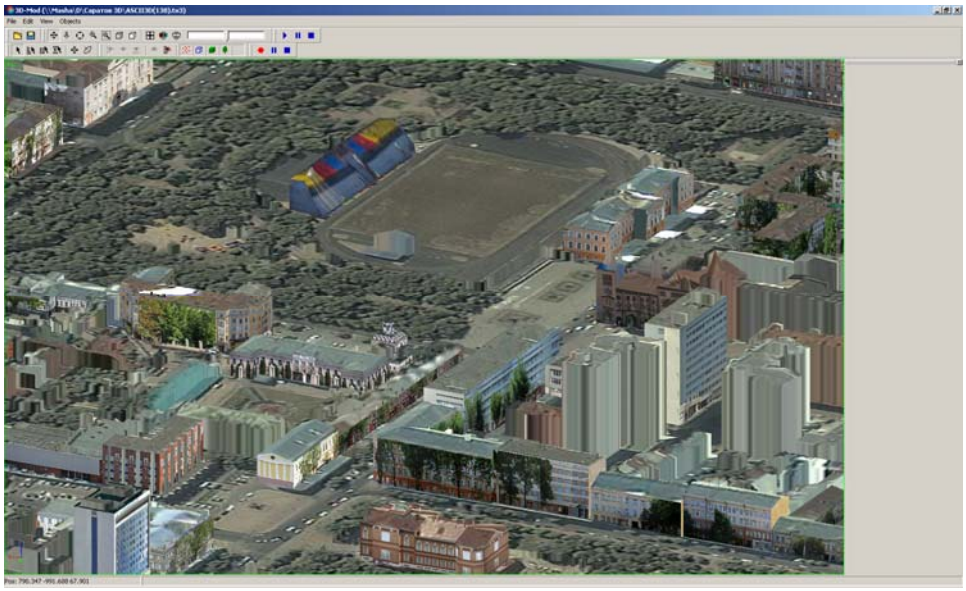
The capability of texture mapping to 3D objects for the most realistic visualization: possibility of making necessary shading and/or illusion of relief is implemented in the 3D Mod program.

The process of texture mapping to 3D-objects can be divided into the following steps:

- Load or import vector objects in ASCII3D, ASCII or 3DS formats with the *. txt, *. 3ds and *. tx3 extensions, respectively;
- The process of texture mapping includes: loading texture and editing of the texture coordinates;
- Saving of 3D objects with the mapped texture to the 3D Mod native format or export to the necessary data format for further use in other software.

11.2.1. Texture mapping to 3D objects

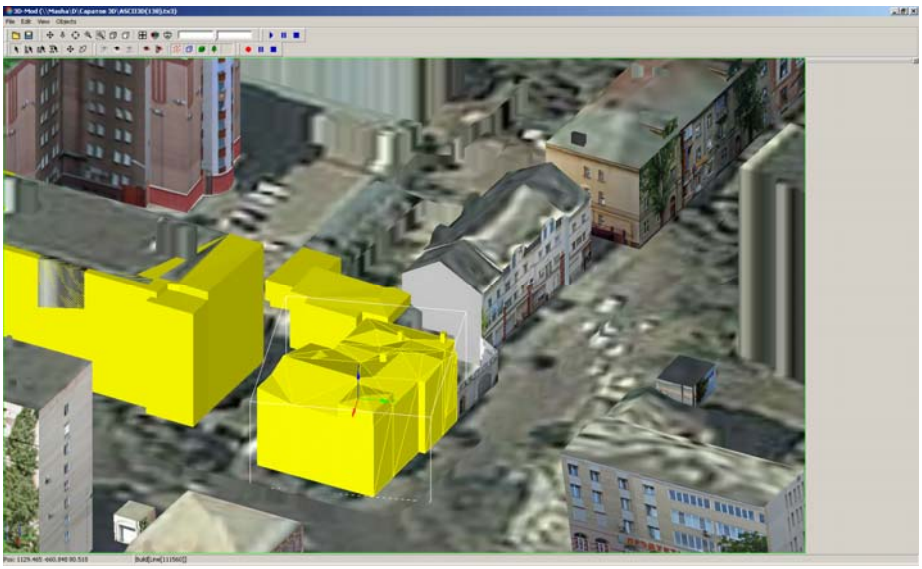
- Load the file of 3D objects in working area using the **File | Open** menu command (the dialog selection of datafile in the 3D Mod native format is opened when the command) or the **File | Import** menu command (the dialog selection of supported formats is opened when the command).



General view of 3D scene

- Set the object in the working area of 3D window in a comfortable view to texture mapping using the **Toolbar** (see chapter **Main menu of 3D Mod program**).

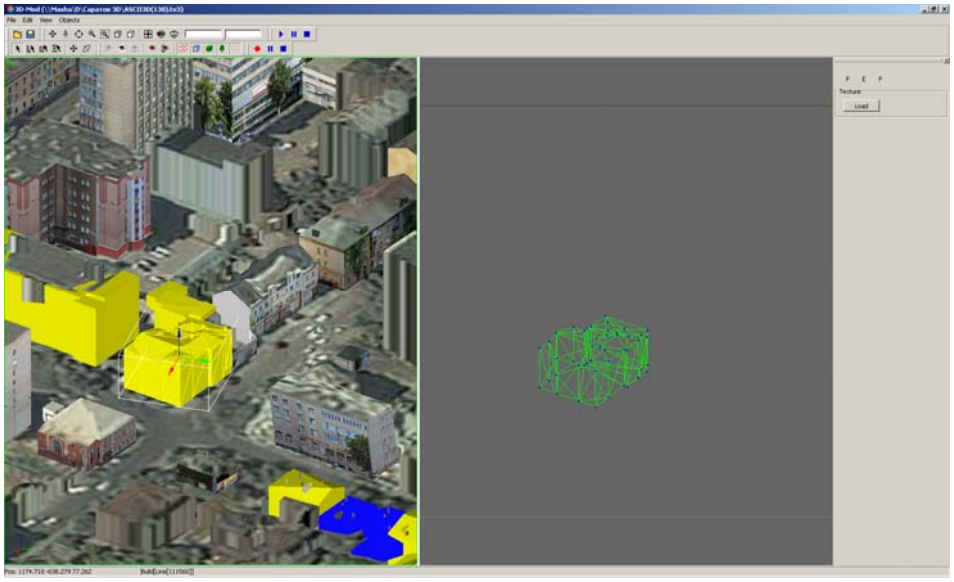
- Select object using the mouse click.



Start working using the **Edit | Edit texture coordinates** menu command in case, if the texture was not initially been mapped to the object.

In that case, if the texture initially has been mapped to object, use the sequence of menu command: **Edit | New material**, **Edit | Edit texture coordinates**. Some objects of 3D scenes can have the common texture. Changing the texture of one of the objects makes it automatically changed for the other objects. To define the unique texture for selected object of 3D scene it's necessary previously to define the new material for it.

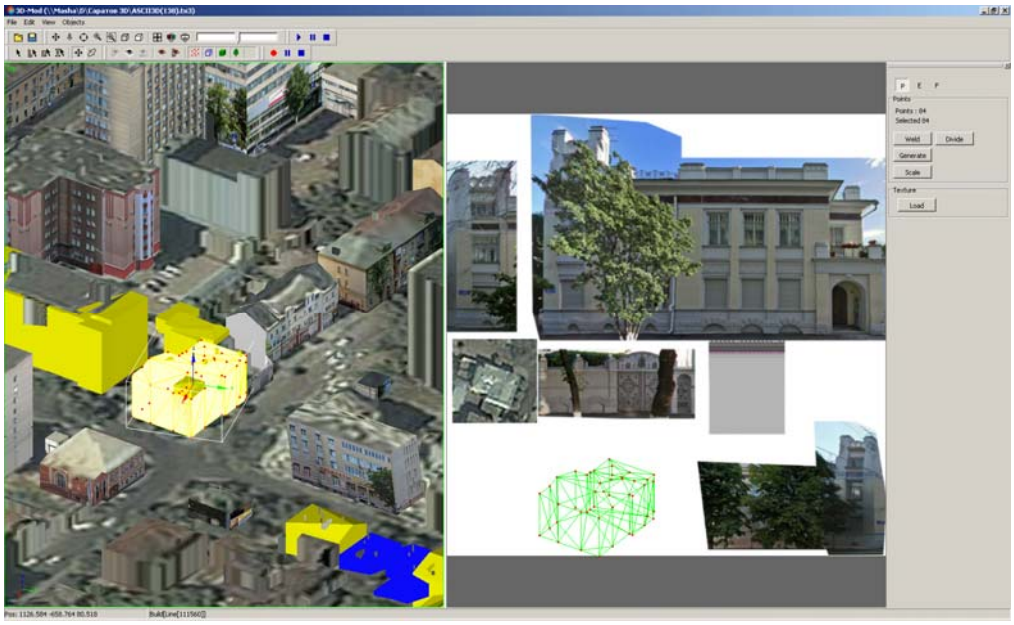
The working area of 3D window divides into 2 parts (3D and 2D).



The control panel of 2D window is on the right side of the screen. The 2D window is used for editing of texture coordinates, loading and displaying of texture, as well as displaying the polygonal frame of object.

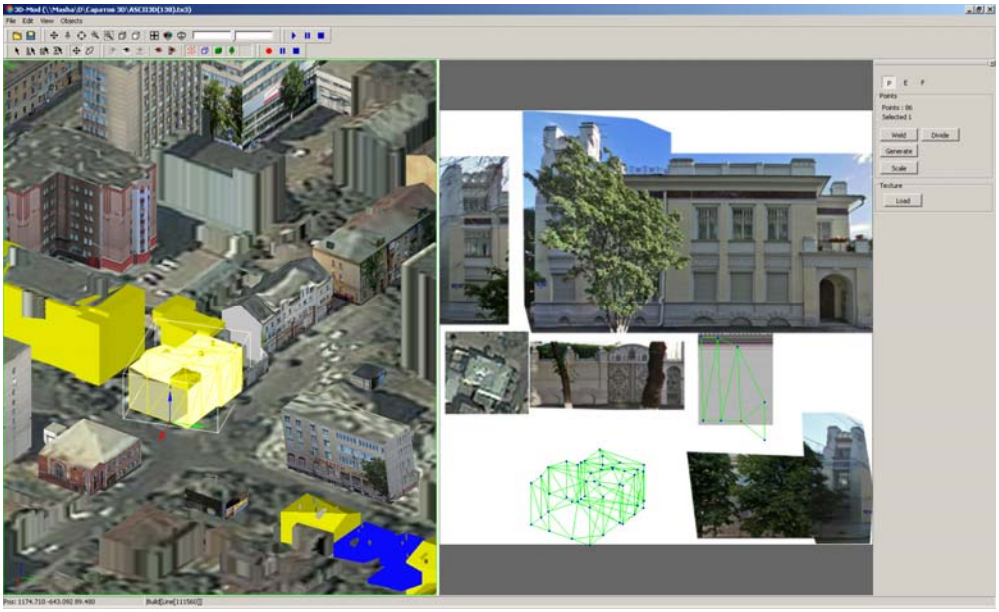
- Load the texture using the

button in the control panel of 2D window. The dialog selection of raster file with *.bmp extension opens. Select necessary file.



- Start the process of the texture mapping by editing texture coordinates. This process is performed by “binding” raster images to the vertices of faces of the polygonal frame. And it is showed in the 3D window in real time, with corresponding vertices, which are displayed simultaneously (in the form of the origin of coordinates icon), for the most correctly mapping of the texture on the object.

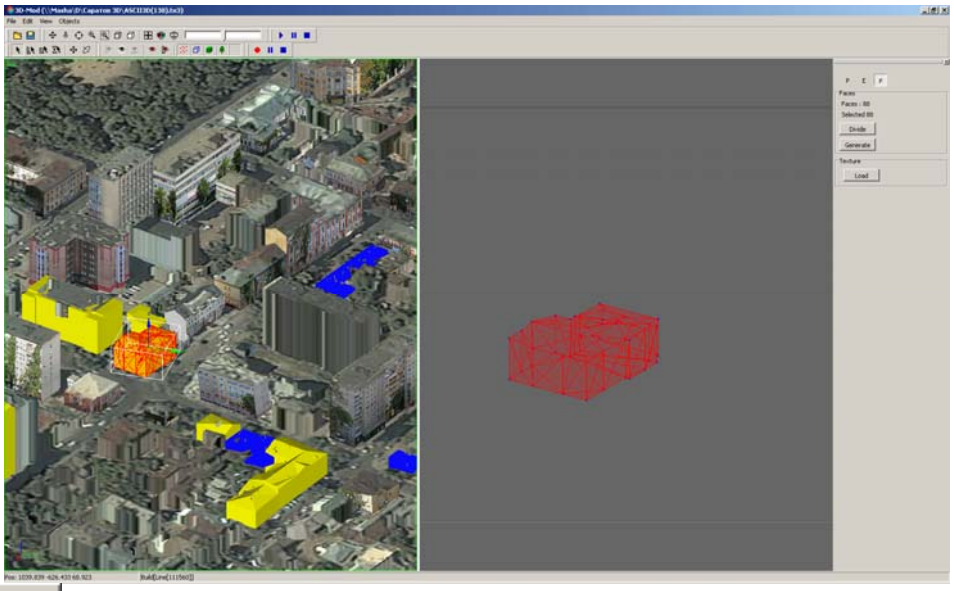
Select the part of object (face) in the F (faces) mode using the left mouse key and **Shift** button and push the Divide button for mapping the selected fragment of texture to corresponding face of 3D object. Push the button (**Move object**) on the main **Toolbar**. Point the selected face of polygonal frame with the marker in 2D window. Drag the face by moving the mouse along with its left button pressed and pull toward to the corresponding raster image.
Then, it is necessary “to bind” the texture to the vertices of selected face of the polygonal frame. Activate the P (point) mode. Push the button (**Move object**) on the main **Toolbar**. Select one of the vertices by clicking and dragging it using the left mouse button, move it to the needed position on a raster image (3D window shows in real time the vertice of a face which a fragment of texture is bound to).



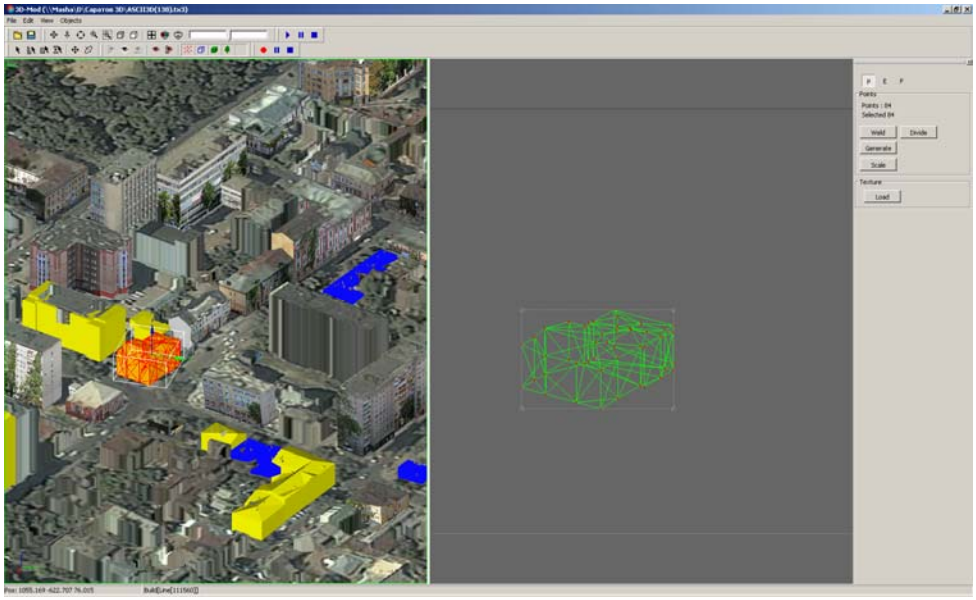
Select the next face of object and continue the process of texture mapping.

During the texture coordinates editing the necessity to change position of edited face in the 2D window may occur to texture mapping correctly. Use the Generate button. Set the editable face in the working area of 3D window in a comfortable view and push the Generate button. Editable face displays in the 2D window in the same view.

Activate the F (faces) mode and push the button (**Move object**) on the main **Toolbar** for the polygonal frame movement in the 2D window. Select the polygonal frame by clicking and dragging it using the left mouse button, point it with the marker, move it to the needed position the same way.



The Scale button is performed to change size of polygonal frame or its faces. Activate the P (point) mode. Select the polygonal frame or face by clicking and dragging it using the left mouse button and push the Scale button. The zoom tool displays around the object. Pull one of the frame corners to change size.



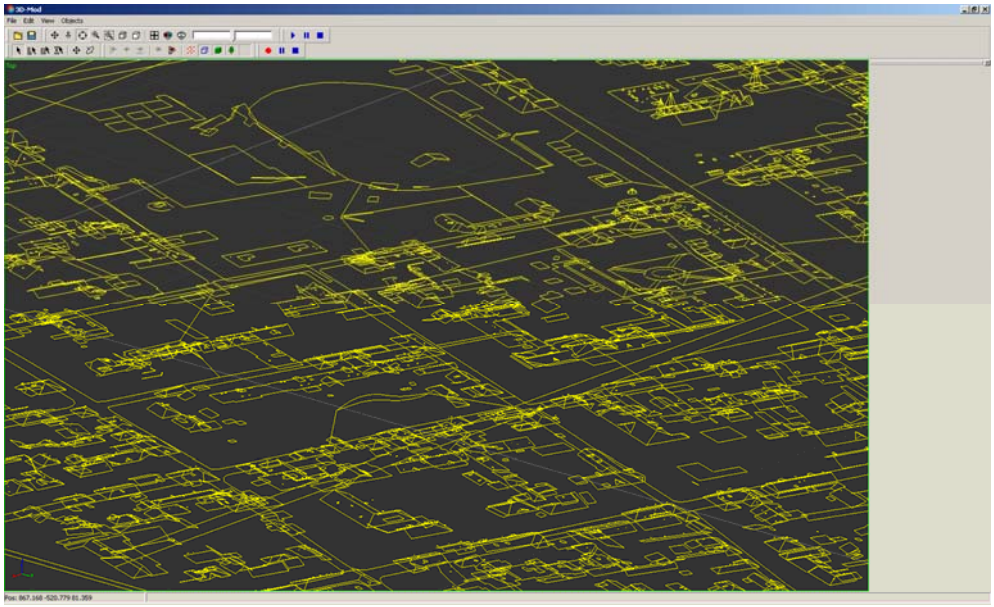
Use the **File | Save** menu command to save the changes in the 3D scene in the 3D Mod native format or the **File | Save as** menu command to save changes in the current file under a new name. Use the **File | Export selected** menu command to save and export changes of selected object to the necessary data format for further use in other software (the dialog selection of supported formats is opened when the command).

11.2.2. Objects editing in 3D Mod

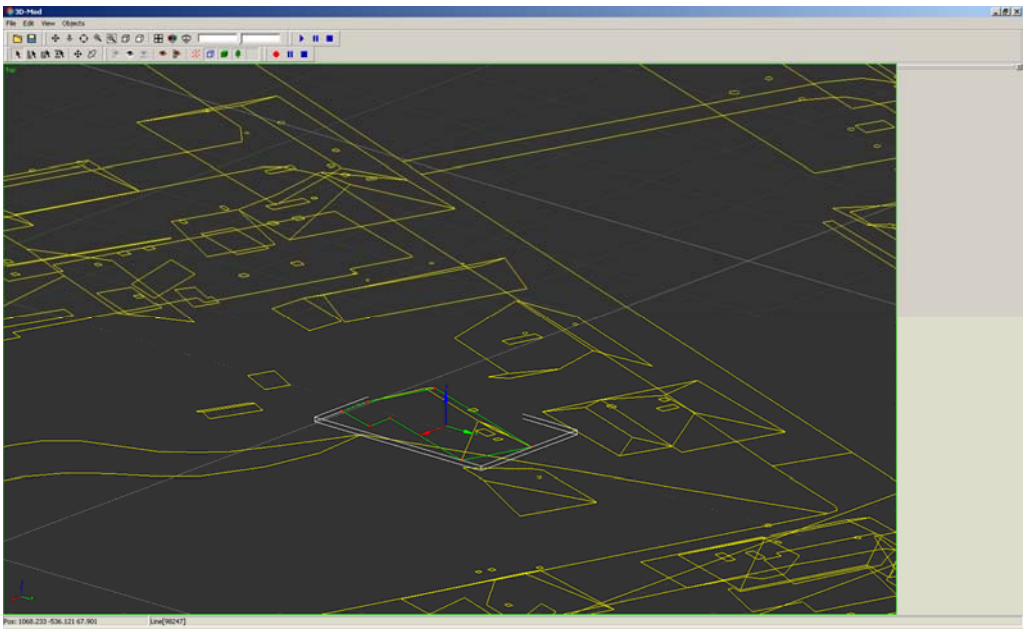
The editing feature in the 3D Mod program is implemented for removing stereo-plotting errors of the initial 2D vector objects and editing of already created 3D objects.

11.2.2.1. *Removing of the stereo-plotting errors of the initial 2D vector objects*

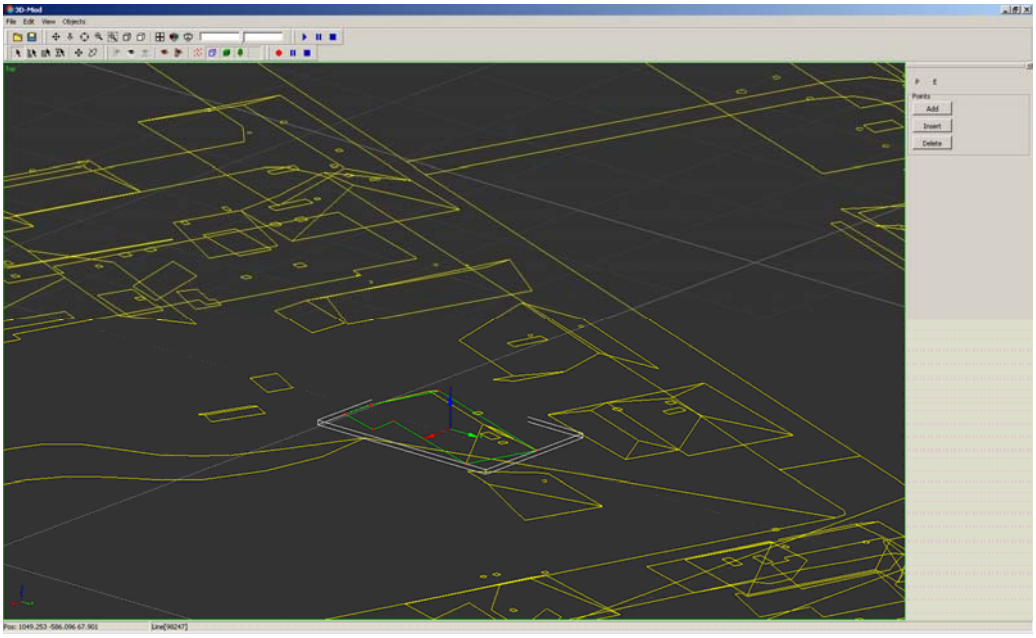
Import the layer of initial 2D vector objects for editing using the **File | Import** menu command.



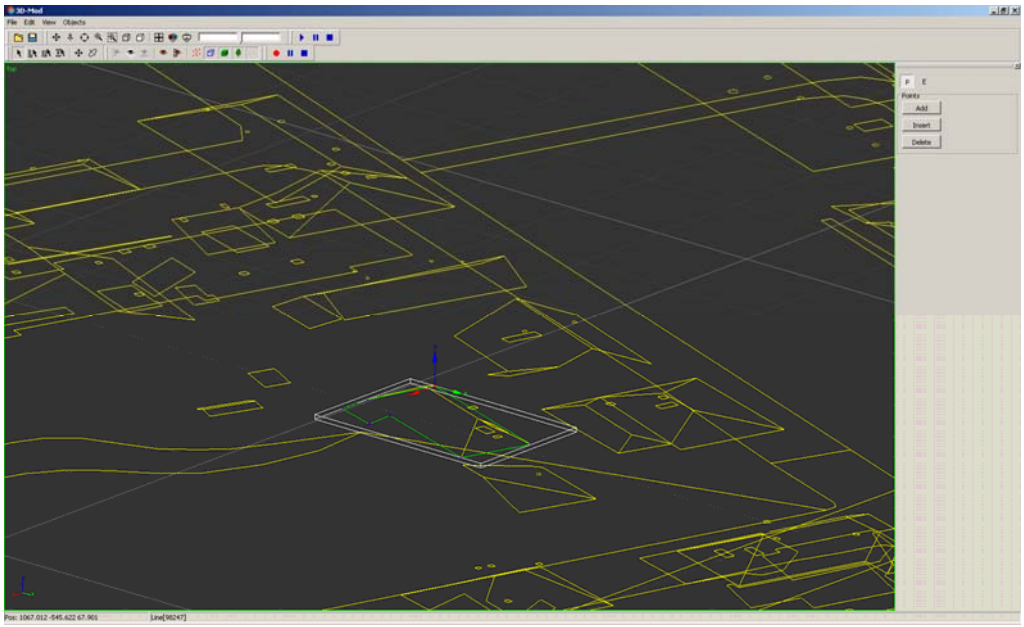
Select the editable object by mouse clicking.



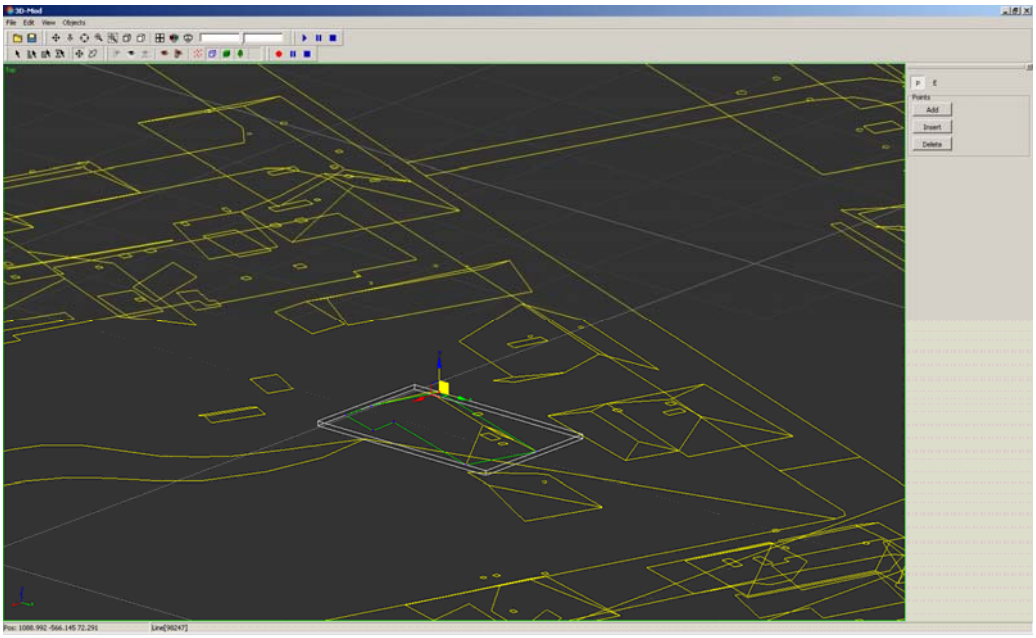
Use the **Edit | Edit** menu command.



Activate the P (point) mode and push the button (**Move object**) on the main **Toolbar** for removing of stereo-plotting errors.



Point the reference system origin icon with the marker the way that one of the planes is highlighted in yellow. Pull it using the left mouse button and edit the object.



Use the **Objects | Build** menu command for creating 3D object from edited vectors.

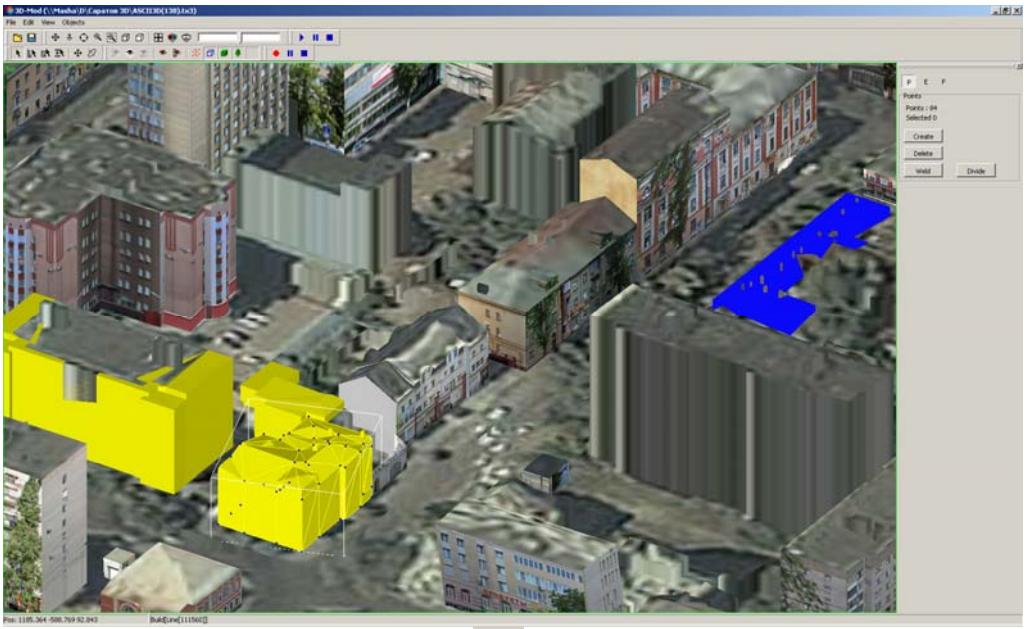
Use the **File | Save** menu command to save the changes in the 3D scene in the 3D Mod native format or the **File | Save as** menu command to save changes in the current file under a new name. Use the **File | Export selected** menu command to save and export changes of selected object to the necessary data format for further use in other software (the dialog selection of supported formats is opened when the command) or the **File | Export** menu command to save and export changes from the whole 2D objects layer.

© 2011

11.2.2.2. *Editing of 3D objects*

Open the necessary 3D object or scene for editing using the **File | Open** menu command. Select the editable object by mouse clicking. Use the **Edit | Edit** menu command.

For 3D objects vertices editing:



Activate the P (point) mode and push the button (**Move object**) on the main **Toolbar**. Select the appropriate vertex to edit. Point the reference system origin icon with the marker the way that one of the planes is highlighted in yellow. Pull it using the left mouse button and edit the position of vertex.

Select the vertex and push the

button to delete.

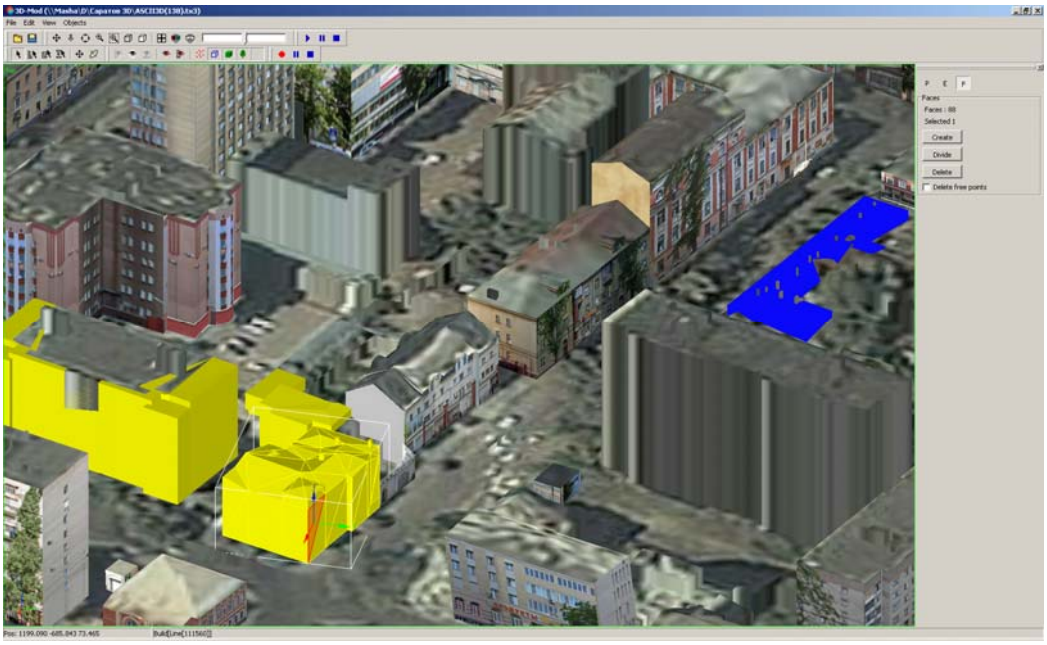
Select the vertices and push the

button to join topologycally the several vertices.

Use the

button to divide vertices.

For 3D objects faces editing:



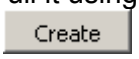
Activate the F (faces) mode and push the button (**Move object**) on the main **Toolbar**. Select the appropriate face to edit. Point the reference system origin icon with the marker the way that one of the planes is highlighted in yellow. Pull it using the left mouse button and edit the position of face.

Select the face and push the

button to delete.

Select the face and push the

button to split the 3D object into faces. Point the reference system origin icon with the marker the way that one of the planes is highlighted in yellow. Pull it using the left mouse button and detach the face.

Use the 

button to create the face. Point the first vertex of face being created with the marker. Create the face, successively selecting three vertices counterclockwise.
The **Delete free points** check box removes the vertices that are not attached with any faces.

Note: The front side of face defined by a sequence of vertices counterclockwise.

Use the **File | Save** menu command to save the changes in the 3D scene in the 3D Mod native format or the **File | Save as** menu command to save changes in the current file under a new name. Use the **File | Export selected** menu command to save and export changes of selected object to the necessary data format for further use in other software (the dialog selection of supported formats is opened when the command) or **File | Export** menu command to save and export changes from the whole 2D objects layer.

11.3. *Appendix A. Description of ASCII format*

In this format each vector object is represented as a sequence of vertices described by their 3D coordinates. The file of this format is a sequence of records separated by "*" symbol. Each record describes an object, which can be a point or a polylines.
Each record consists of a type of an object and rows with 3D coordinates of its vertices. If an object is a point, its name (text string) may follow its coordinates. The example of ASCII file is shown below.

Road	- type - text string;
1234.67,4567.67,565.453	- X1,Y1,Z1 - 1-st vertex coordinates;
1245.6,7439.570,860.958	- X2,Y2,Z2 - 2-nd vertex coordinates;
...	- ...
*	- separator of object descriptions;
Point	- type - text string;
1257.85,2198.76,459.56,Point1	- X, Y, Z - point coordinates and name (arbitrary text);
*	- ...
...	- ...

11.4. *Appendix B. Description of ASCII-A format*

In addition to vertices coordinates the extended ASCII-A format contains the following information: an object type, a layer number, names and values of an object attributes. The structure of this format file is mainly the same as of ASCII format file. But the header of the file contains several lines, describing an object type and attributes.

Below is an example of an object description section with explanation:
L 101 1 15 6
OBJECT_NAME=Highway
OBJECT_COLOR=3
OBJECT_SYMBOL=R

OBJECT_SIZE=5.5
WIDTH=7.2
ROADNAME= Highway_1234
545566.505,473671.817,77.850
545715.103,473656.072,78.310
545782.001,473567.393,78.156
545860.428,473463.139,77.974
545847.506,473339.305,77.380
545795.032,473249.288,76.795
545517.126,473365.500,76.318
545269.605,473463.426,75.869
*

The first line of an object description has the following structure:
Type Code Layer N1 N2,
where:

- **Type** - the symbol describing an object type (L, P, C, R);
- **Code** - the code of an object;
- **Layer** - the layer number;
- **N1** - the total number of lines of an object description;
- **N2** - the number of lines describing object attributes.

Then lines with descriptions of attributes follow. An attribute description line looks like:
Name=Value
There are four following attributes with reserved names:

- OBJECT_NAME – name of object;
- OBJECT_COLOR – number of object color;
- OBJECT_SYMBOL – symbol assigned to object;
- OBJECT_SIZE – size of object.

During import values of these attributes are assigned to the **Name**, **Color**, **Symbol** and **Size** properties respectively.
Lines with an object attributes description are followed by the lines with coordinates of object vertices. The coordinate description lines have the same format as described above for the ASCII format without attributes. Groups of lines corresponding to objects descriptions are separated by lines with the "*" symbol, but unlike the previous case, these lines are not obligatory because the beginning of an object description is recognized by its first line.
The main role in recognizing of the beginning of objects description plays the first string.

In addition to standard some attributes are used during 3D-objects creation. These objects have the following default names and appropriate function:

object=
- contour or line with this attribute is bounding contour;

break=
- contour, line or point with this attribute is break line;

library=
- point with this attribute specifies a height of library object;

none=
- contour, line or point with this attribute is not used for objects creation.